ANSYS FLUENT User's Guide

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 13.0
November 2010

ANSYS, Inc. is
certified to ISO
9001:2008.

Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Using This Manual lxi
1. The Contents of This Manual lxi
2. The Contents of the Other Manuals lxiii
3. Typographical Conventions lxiii
4. Mathematical Conventions lxiv
5. Technical Support lxv
1. Starting and Executing ANSYS FLUENT 1
1.1. Starting ANSYS FLUENT 1
1.1.1. Single-Precision and Double-Precision Solvers 1
1.1.2. Starting ANSYS FLUENT Using FLUENT Launcher 2
1.1.2.1. Setting General Options in FLUENT Launcher 3
1.1.2.2. Setting Parallel Options in FLUENT Launcher 5
1.1.2.3. Setting Remote Options in FLUENT Launcher 6
1.1.2.4. Setting Scheduler Options in FLUENT Launcher 7
1.1.2.5. Setting Environment Options in FLUENT Launcher 8
1.1.3. Starting ANSYS FLUENT on a Windows System 9
1.1.4. Starting ANSYS FLUENT on a Linux/UNIX System 10
1.1.5. Command Line Startup Options 10
1.1.5.1. Graphics Options 11
1.1.5.2. Parallel Options 12
1.1.5.3. Postprocessing Option 12
1.1.5.4. SGE Options 13
1.1.5.5. LSF Options 13
1.1.5.6. Version and Release Options 13
1.1.5.7. Other Startup Options 13
1.1.6. Remote Simulation Facility (RSF) 13
1.2. Running ANSYS FLUENT in Batch Mode 14
1.2.1. Background Execution on Linux/UNIX Systems 14
1.2.2. Background Execution on Windows Systems 15
1.2.3. Batch Execution Options 16
1.3. Checkpointing an ANSYS FLUENT Simulation 17
1.4. Cleaning Up Processes From an ANSYS FLUENT Simulation 18
1.5. Exiting the Program 19
2. Graphical User Interface (GUI) 21
2.1. GUI Components 21
2.1.1. The Menu Bar 22
2.1.2. Toolbars 22
2.1.2.1. The Standard Toolbar 23
2.1.2.2. The Graphics Toolbar 23
2.1.3. The Navigation Pane 24
2.1.4. Task Pages 26
2.1.5. The Console 26
2.1.6. Dialog Boxes 27
2.1.6.1. Input Controls 28
2.1.6.1.1. Tabs 28
2.1.6.1.2. Buttons 28
2.1.6.1.3. Check Boxes 28
2.1.6.1.4. Radio Buttons 29
2.1.6.1.5. Text Entry Boxes 29
2.1.6.1.6. Integer Number Entry Boxes 29

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.6.1.7. Real Number Entry Boxes ... 29
2.1.6.1.8. Single-Selection Lists ... 30
2.1.6.1.9. Multiple-Selection Lists ... 30
2.1.6.1.10. Drop-Down Lists ... 30
2.1.6.1.11. Scales ... 31
2.1.6.1.11.1. Types of Dialog Boxes ... 31
2.1.6.1.12. Information Dialog Boxes ... 31
2.1.6.1.13. Warning Dialog Boxes ... 32
2.1.6.1.14. Error Dialog Boxes ... 32
2.1.6.1.15. The Working Dialog Box ... 32
2.1.6.1.16. Question Dialog Box ... 32
2.1.6.1.17. The **Select File** Dialog Box ... 33
2.1.6.1.17.1. The **Select File** Dialog Box (Windows) ... 33
2.1.6.1.17.2. The Select File Dialog Box (UNIX or Linux) ... 33
2.1.7. Graphics Windows ... 36
2.1.7.1. Printing the Contents of the Graphics Window (Windows Systems Only) ... 38
2.1.7.2. Using the **Page Setup** Dialog Box (Windows Systems Only) ... 38
2.2. Customizing the Graphical User Interface (UNIX Systems Only) ... 40
2.3. Using the GUI Help System ... 41
2.3.1. Task Page and Dialog Box Help ... 42
2.3.2. Context-Sensitive Help (UNIX Only) ... 42
2.3.3. Opening the User's Guide Table of Contents ... 42
2.3.4. Opening the User's Guide Index ... 43
2.3.5. Opening the Reference Guide ... 43
2.3.6. Help on Help ... 44
2.3.7. Accessing Printable Manuals ... 44
2.3.8. Help for Text Interface Commands ... 44
2.3.9. Accessing the Online Technical Support Web Site ... 44
2.3.10. Obtaining License Use Information ... 44
2.3.11. Version and Release Information ... 45
3. Text User Interface (TUI) ... 47
3.1. Text Menu System ... 47
3.1.1. Command Abbreviation ... 48
3.1.2. Command Line History ... 48
3.1.3. Scheme Evaluation ... 49
3.1.4. Aliases ... 49
3.2. Text Prompt System ... 50
3.2.1. Numbers ... 50
3.2.2. Booleans ... 50
3.2.3. Strings ... 51
3.2.4. Symbols ... 51
3.2.5. Filenames ... 51
3.2.6. Lists ... 51
3.2.7. Evaluation ... 53
3.2.8. Default Value Binding ... 53
3.3. Interrupts ... 53
3.4. System Commands ... 53
3.4.1. System Commands for UNIX-based Operating Systems ... 53
3.4.2. System Commands for Windows Operating Systems ... 54
3.5. Text Menu Input from Character Strings ... 55
3.6. Using the Text Interface Help System ... 55
4. Reading and Writing Files ... 57

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1. Shortcuts for Reading and Writing Files ... 57
4.1.1. Default File Suffixes ... 58
4.1.2. Binary Files ... 59
4.1.3. Detecting File Format ... 59
4.1.4. Recent File List ... 59
4.1.5. Reading and Writing Compressed Files ... 59
4.1.5.1. Reading Compressed Files ... 59
4.1.5.2. Writing Compressed Files ... 60
4.1.6. Tilde Expansion (Linux/UNIX Systems Only) ... 60
4.1.7. Automatic Numbering of Files ... 61
4.1.8. Disabling the Overwrite Confirmation Prompt ... 61
4.1.9. Toolbar Buttons ... 61
4.2. Reading Mesh Files ... 62
4.2.1. Reading TGrid Mesh Files ... 62
4.2.2. Reading Surface Meshes ... 63
4.2.3. Reading GAMBIT and GeoMesh Mesh Files ... 63
4.2.4. Reading PreBFC Unstructured Mesh Files ... 63
4.3. Reading and Writing Case and Data Files ... 63
4.3.1. Reading and Writing Case Files ... 64
4.3.2. Reading and Writing Data Files ... 64
4.3.3. Reading and Writing Case and Data Files Together ... 65
4.3.4. Automatic Saving of Case and Data Files ... 65
4.4. Reading and Writing Parallel Data Files ... 68
4.4.1. Writing Parallel Data Files ... 68
4.4.2. Reading Parallel Data Files ... 69
4.4.3. Availability and Limitations ... 69
4.5. Reading FLUENT/UNS and RAMPANT Case and Data Files ... 70
4.6. Reading and Writing Profile Files ... 70
4.6.1. Reading Profile Files ... 70
4.6.2. Writing Profile Files ... 71
4.7. Reading and Writing Boundary Conditions ... 72
4.8. Writing a Boundary Mesh ... 72
4.9. Reading Scheme Source Files ... 73
4.10. Creating and Reading Journal Files ... 73
4.10.1. Procedure ... 74
4.11. Creating Transcript Files ... 75
4.12. Importing Files ... 75
4.12.1. ABAQUS Files ... 77
4.12.2. ANSYS CFX Files ... 77
4.12.3. Meshes and Data in CGNS Format ... 78
4.12.4. EnSight Files ... 78
4.12.5. ANSYS FIDAP Neutral Files ... 79
4.12.6. GAMBIT and GeoMesh Mesh Files ... 79
4.12.7. HYPERMESH ASCII Files ... 79
4.12.8. IC3M Files ... 79
4.12.9. I-deas Universal Files ... 80
4.12.10. LSTC Files ... 80
4.12.11. Marc POST Files ... 81
4.12.12. Mechanical APDL Files ... 81
4.12.13. NASTRAN Files ... 81
4.12.14. PATRAN Neutral Files ... 82
4.12.15. PLOT3D Files ... 82

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.12.16. PTC Mechanica Design Files ... 82
4.12.17. Tecplot Files ... 83
4.12.18. FLUENT 4 Case Files ... 83
4.12.19. PreBFC Files ... 83
4.12.20. Partition Files ... 83
4.12.21. CHEMKIN Mechanism ... 84
4.13. Exporting Solution Data ... 84
4.13.1. Exporting Limitations ... 84
4.14. Exporting Solution Data after a Calculation ... 85
4.14.1. ABAQUS Files ... 87
4.14.2. Mechanical APDL Files ... 87
4.14.3. Mechanical APDL Input Files ... 88
4.14.4. ASCII Files ... 89
4.14.5. AVS Files ... 89
4.14.6. ANSYS CFD-Post-Compatible Files ... 89
4.14.7. CGNS Files ... 90
4.14.8. Data Explorer Files ... 90
4.14.9. EnSight Case Gold Files ... 91
4.14.10. FAST Files ... 93
4.14.11. FAST Solution Files ... 93
4.14.12. Fieldview Unstructured Files ... 94
4.14.13. I-deas Universal Files ... 94
4.14.14. NASTRAN Files ... 95
4.14.15. PATRAN Files ... 95
4.14.16. RadTherm Files ... 96
4.14.17. Tecplot Files ... 96
4.15. Exporting Steady-State Particle History Data ... 97
4.16. Exporting Data During a Transient Calculation ... 98
4.16.1. Creating Automatic Export Definitions for Solution Data ... 100
4.16.2. Creating Automatic Export Definitions for Transient Particle History Data ... 102
4.17. Exporting to ANSYS CFD-Post ... 104
4.18. Managing Solution Files ... 106
4.19. Mesh-to-Mesh Solution Interpolation ... 106
4.19.1. Performing Mesh-to-Mesh Solution Interpolation ... 107
4.19.2. Format of the Interpolation File ... 108
4.20. Mapping Electromagnetic Losses ... 109
4.21. Mapping Data for Fluid-Structure Interaction (FSI) Applications ... 113
4.21.1. FEA File Formats ... 113
4.21.2. Using the FSI Mapping Dialog Boxes ... 113
4.22. Saving Picture Files ... 117
4.22.1. Using the Save Picture Dialog Box ... 118
4.22.1.1. Choosing the Picture File Format ... 119
4.22.1.2. Specifying the Color Mode ... 119
4.22.1.3. Choosing the File Type ... 120
4.22.1.4. Defining the Resolution ... 120
4.22.1.5. Picture Options ... 120
4.22.2. Picture Options for PostScript Files ... 120
4.22.2.1. Window Dumps (Linux/UNIX Systems Only) ... 121
4.22.2.2. Previewing the Picture Image ... 121
4.23. Setting Data File Quantities ... 121
4.24. The .fluent File ... 123
5. Unit Systems ... 125

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1. Restrictions on Units ... 125
5.2. Units in Mesh Files ... 126
5.3. Built-In Unit Systems in ANSYS FLUENT ... 126
5.4. Customizing Units ... 126
5.4.1. Listing Current Units ... 127
5.4.2. Changing the Units for a Quantity ... 127
5.4.3. Defining a New Unit ... 127
5.4.3.1. Determining the Conversion Factor ... 128
6. Reading and Manipulating Meshes ... 129
6.1. Mesh Topologies ... 129
6.1.1. Examples of Acceptable Mesh Topologies ... 131
6.1.2. Face-Node Connectivity in ANSYS FLUENT ... 136
6.1.2.1. Face-Node Connectivity for Triangular Cells ... 137
6.1.2.2. Face-Node Connectivity for Quadrilateral Cells ... 137
6.1.2.3. Face-Node Connectivity for Tetrahedral Cells ... 138
6.1.2.4. Face-Node Connectivity for Wedge Cells ... 139
6.1.2.5. Face-Node Connectivity for Pyramidal Cells ... 140
6.1.2.6. Face-Node Connectivity for Hex Cells ... 140
6.1.2.7. Face-Node Connectivity for Polyhedral Cells ... 141
6.1.3. Choosing the Appropriate Mesh Type ... 141
6.1.3.1. Setup Time ... 141
6.1.3.2. Computational Expense ... 142
6.1.3.3. Numerical Diffusion ... 142
6.2. Mesh Requirements and Considerations ... 143
6.2.1. Geometry/Mesh Requirements ... 143
6.2.2. Mesh Quality ... 144
6.2.2.1. Mesh Element Distribution ... 146
6.2.2.2. Cell Quality ... 147
6.2.2.3. Smoothness ... 148
6.2.2.4. Flow-Field Dependency ... 148
6.3. Mesh Import ... 148
6.3.1. GAMBIT Mesh Files ... 148
6.3.2. GeoMesh Mesh Files ... 149
6.3.3. TGrid Mesh Files ... 149
6.3.4. PreBFC Mesh Files ... 149
6.3.4.1. Structured Mesh Files ... 149
6.3.4.2. Unstructured Triangular and Tetrahedral Mesh Files ... 149
6.3.5. ICEM CFD Mesh Files ... 150
6.3.6. I-deas Universal Files ... 150
6.3.6.1. Recognized I-deas Datasets ... 150
6.3.6.2. Grouping Nodes to Create Face Zones ... 150
6.3.6.3. Grouping Elements to Create Cell Zones ... 151
6.3.6.4. Deleting Duplicate Nodes ... 151
6.3.7. NASTRAN Files ... 151
6.3.7.1. Recognized NASTRAN Bulk Data Entries ... 151
6.3.7.2. Deleting Duplicate Nodes ... 152
6.3.8. PATRAN Neutral Files ... 152
6.3.8.1. Recognized PATRAN Datasets ... 152
6.3.8.2. Grouping Elements to Create Cell Zones ... 153
6.3.9. Mechanical APDL Files ... 153
6.3.9.1. Recognized ANSYS 5.4 and 5.5 Datasets ... 153
6.3.10. ANSYS CFX Files ... 154

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.11. Using the fe2ram Filter to Convert Files ... 155
6.3.12. Using the tpoly Filter to Remove Hanging Nodes ... 155
6.3.12.1. Limitations ... 156
6.3.13. FLUENT/UNS and RAMPANT Case Files ... 156
6.3.14. FLUENT 4 Case Files ... 156
6.3.15. ANSYS FIDAP Neutral Files ... 157
6.3.16. Reading Multiple Mesh/Case/Data Files ... 157
6.3.16.1. Using ANSYS FLUENT's Ability to Read Multiple Mesh Files ... 157
6.3.16.2. Using TGrid or tmerge ... 159
6.3.17. Reading Surface Mesh Files ... 160
6.4. Non-Conformal Meshes ... 161
6.4.1. Non-Conformal Mesh Calculations ... 161
6.4.1.1. The Periodic Boundary Condition Option ... 163
6.4.1.2. The Periodic Repeats Option ... 164
6.4.1.3. The Coupled Wall Option ... 166
6.4.2. Non-Conformal Interface Algorithm ... 167
6.4.3. Requirements and Limitations of Non-Conformal Meshes ... 168
6.4.4. Using a Non-Conformal Mesh in ANSYS FLUENT ... 169
6.5. Checking the Mesh ... 172
6.5.1. Mesh Check Report ... 173
6.5.2. Repairing Meshes ... 174
6.6. Reporting Mesh Statistics ... 175
6.6.1. Mesh Size ... 175
6.6.2. Memory Usage ... 176
6.6.2.1. UNIX Systems ... 176
6.6.2.2. Windows Systems ... 176
6.6.3. Mesh Zone Information ... 177
6.6.4. Partition Statistics ... 177
6.7. Converting the Mesh to a Polyhedral Mesh ... 177
6.7.1. Converting the Domain to a Polyhedra ... 178
6.7.1.1. Limitations ... 181
6.7.2. Converting Skewed Cells to Polyhedra ... 182
6.7.2.1. Limitations ... 182
6.7.2.2. Using the Convert Skewed Cells Dialog Box ... 182
6.8. Modifying the Mesh ... 183
6.8.1. Merging Zones ... 184
6.8.1.1. When to Merge Zones ... 184
6.8.1.2. Using the Merge Zones Dialog Box ... 185
6.8.2. Separating Zones ... 185
6.8.2.1. Separating Face Zones ... 186
6.8.2.1.1. Methods for Separating Face Zones ... 186
6.8.2.1.2. Inputs for Separating Face Zones ... 186
6.8.2.2. Separating Cell Zones ... 188
6.8.2.2.1. Methods for Separating Cell Zones ... 188
6.8.2.2.2. Inputs for Separating Cell Zones ... 188
6.8.3. Fusing Face Zones ... 190
6.8.3.1. Inputs for Fusing Face Zones ... 190
6.8.3.1.1. Fusing Zones on Branch Cuts ... 191
6.8.4. Creating Conformal Periodic Zones ... 191
6.8.5. Slitting Periodic Zones ... 192
6.8.6. Slitting Face Zones ... 193
6.8.6.1. Inputs for Slitting Face Zones ... 193

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8.7. Orienting Face Zones ... 194
6.8.8. Extruding Face Zones ... 194
6.8.8.1. Specifying Extrusion by Displacement Distances ... 194
6.8.8.2. Specifying Extrusion by Parametric Coordinates ... 194
6.8.9. Replacing, Deleting, Deactivating, and Activating Zones ... 195
6.8.9.1. Replacing Zones ... 195
6.8.9.2. Deleting Zones ... 196
6.8.9.3. Deactivating Zones ... 196
6.8.9.4. Activating Zones ... 197
6.8.. Replacing the Mesh ... 198
6.8..1. Inputs for Replacing the Mesh ... 198
6.8..2. Limitations ... 199
6.8.11. Reordering the Domain and Zones ... 199
6.8.11.1. About Reordering ... 200
6.8.12. Scaling the Mesh ... 200
6.8.12.1. Using the Scale Mesh Dialog Box ... 201
6.8.12.1.1. Changing the Unit of Length ... 201
6.8.12.1.2. Unscaling the Mesh ... 201
6.8.12.1.3. Changing the Physical Size of the Mesh ... 202
6.8.13. Translating the Mesh ... 202
6.8.13.1. Using the Translate Mesh Dialog Box ... 202
6.8.14. Rotating the Mesh ... 203
6.8.14.1. Using the Rotate Mesh Dialog Box ... 203
7. Cell Zone and Boundary Conditions ... 205
7.1. Overview ... 205
7.1.1. Available Cell Zone and Boundary Types ... 205
7.1.2. The Cell Zone and Boundary Conditions Task Page ... 206
7.1.3. Changing Cell and Boundary Zone Types ... 207
7.1.3.1. Categories of Zone Types ... 207
7.1.4. Setting Cell Zone and Boundary Conditions ... 208
7.1.5. Copying Cell Zone and Boundary Conditions ... 208
7.1.6. Changing Cell or Boundary Zone Names ... 209
7.1.7. Defining Non-Uniform Cell Zones and Boundary Conditions ... 210
7.1.8. Defining and Viewing Parameters ... 210
7.1.8.1. Creating a New Parameter ... 211
7.1.9. Selecting Cell or Boundary Zones in the Graphics Display ... 213
7.1.10. Operating and Periodic Conditions ... 213
7.1.11. Highlighting Selected Boundary Zones ... 213
7.1.12. Saving and Reusing Cell Zone and Boundary Conditions ... 214
7.2. Cell Zone Conditions ... 214
7.2.1. Fluid Conditions ... 214
7.2.1.1. Inputs for Fluid Zones ... 215
7.2.1.1.1. Defining the Fluid Material ... 216
7.2.1.1.2. Defining Sources ... 216
7.2.1.1.3. Defining Fixed Values ... 216
7.2.1.1.4. Specifying a Laminar Zone ... 216
7.2.1.1.5. Specifying a Reaction Mechanism ... 216
7.2.1.1.6. Specifying the Rotation Axis ... 217
7.2.1.1.7. Defining Zone Motion ... 217
7.2.1.1.8. Defining Radiation Parameters ... 218
7.2.2. Solid Conditions ... 218
7.2.2.1. Inputs for Solid Zones ... 218

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.2.1.1. Defining the Solid Material ... 219
7.2.2.1.2. Defining a Heat Source ... 219
7.2.2.1.3. Defining a Fixed Temperature ... 219
7.2.2.1.4. Specifying the Rotation Axis ... 219
7.2.2.1.5. Defining Zone Motion ... 220
7.2.2.1.6. Defining Radiation Parameters ... 221
7.2.3. Porous Media Conditions ... 221
7.2.3.1. Limitations and Assumptions of the Porous Media Model ... 221
7.2.3.2. Momentum Equations for Porous Media ... 222
7.2.3.2.1. Darcy's Law in Porous Media ... 223
7.2.3.2.2. Inertial Losses in Porous Media ... 223
7.2.3.3. Treatment of the Energy Equation in Porous Media ... 224
7.2.3.3.1. Effective Conductivity in the Porous Medium ... 224
7.2.3.4. Treatment of Turbulence in Porous Media ... 225
7.2.3.5. Effect of Porosity on Transient Scalar Equations ... 225
7.2.3.6. User Inputs for Porous Media ... 225
7.2.3.6.1. Defining the Porous Zone ... 227
7.2.3.6.2. Defining the Porous Velocity Formulation ... 227
7.2.3.6.3. Defining the Fluid Passing Through the Porous Medium ... 227
7.2.3.6.4. Enabling Reactions in a Porous Zone ... 228
7.2.3.6.5. Including the Relative Velocity Resistance Formulation ... 228
7.2.3.6.6. Defining the Viscous and Inertial Resistance Coefficients ... 228
7.2.3.6.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss ... 231
7.2.3.6.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed ... 232
7.2.3.6.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate ... 233
7.2.3.6.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat ... 234
7.2.3.6.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data ... 234
7.2.3.6.12. Using the Power-Law Model ... 235
7.2.3.6.13. Defining Porosity ... 236
7.2.3.6.14. Defining the Porous Material ... 236
7.2.3.6.15. Defining Sources ... 236
7.2.3.6.16. Defining Fixed Values ... 236
7.2.3.6.17. Suppressing the Turbulent Viscosity in the Porous Region ... 237
7.2.3.6.18. Specifying the Rotation Axis and Defining Zone Motion ... 237
7.2.3.7. Modeling Porous Media Based on Physical Velocity ... 237
7.2.3.7.1. Single Phase Porous Media ... 237
7.2.3.7.2. Multiphase Porous Media ... 238
7.2.3.7.2.1. The Continuity Equation ... 239
7.2.3.7.2.2. The Momentum Equation ... 239
7.2.3.7.2.3. The Energy Equation ... 239
7.2.3.8. Solution Strategies for Porous Media ... 240
7.2.3.9. Postprocessing for Porous Media ... 240
7.2.4. Fixing the Values of Variables ... 241
7.2.4.1. Overview of Fixing the Value of a Variable ... 241
7.2.4.1.1. Variables That Can Be Fixed ... 242
7.2.4.2. Procedure for Fixing Values of Variables in a Zone ... 242
7.2.4.2.1. Fixing Velocity Components ... 243
7.2.4.2.2. Fixing Temperature and Enthalpy ... 243

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.4.2.3. Fixing Species Mass Fractions ... 244
7.2.4.2.4. Fixing Turbulence Quantities ... 244
7.2.4.2.5. Fixing User-Defined Scalars ... 244
7.2.5. Defining Mass, Momentum, Energy, and Other Sources ... 244
7.2.5.1. Sign Conventions and Units ... 245
7.2.5.2. Procedure for Defining Sources ... 245
7.2.5.2.1. Mass Sources ... 246
7.2.5.2.2. Momentum Sources ... 246
7.2.5.2.3. Energy Sources ... 247
7.2.5.2.4. Turbulence Sources ... 247
7.2.5.2.5. Turbulence Sources for the k-ε Model ... 247
7.2.5.2.6. Turbulence Sources for the Spalart-Allmaras Model ... 247
7.2.5.2.7. Turbulence Sources for the k-ω Model ... 247
7.2.5.2.8. Turbulence Sources for the Reynolds Stress Model ... 247
7.2.5.2.9. Mean Mixture Fraction and Variance Sources ... 248
7.2.5.2.10. P-1 Radiation Sources ... 248
7.2.5.2.11. Progress Variable Sources ... 248
7.2.5.2.12. NO, HCN, and NH3 Sources for the NOx Model ... 248
7.2.5.2.13. User-Defined Scalar (UDS) Sources ... 248
7.3. Boundary Conditions ... 249
7.3.1. Flow Inlet and Exit Boundary Conditions ... 249
7.3.2. Using Flow Boundary Conditions ... 249
7.3.2.1. Determining Turbulence Parameters ... 250
7.3.2.1.1. Specification of Turbulence Quantities Using Profiles ... 250
7.3.2.1.2. Uniform Specification of Turbulence Quantities ... 251
7.3.2.1.3. Turbulence Intensity ... 251
7.3.2.1.4. Turbulence Length Scale and Hydraulic Diameter ... 251
7.3.2.1.5. Turbulent Viscosity Ratio ... 252
7.3.2.1.6. Relationships for Deriving Turbulence Quantities ... 252
7.3.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale ... 252
7.3.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity ... 253
7.3.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale ... 253
7.3.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio ... 253
7.3.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence ... 254
7.3.2.1.12. Estimating Specific Dissipation Rate from a Length Scale ... 254
7.3.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio ... 254
7.3.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy ... 255
7.3.2.1.15. Specifying Inlet Turbulence for LES ... 255
7.3.3. Pressure Inlet Boundary Conditions ... 255
7.3.3.1. Inputs at Pressure Inlet Boundaries ... 255
7.3.3.1.1. Summary ... 255
7.3.3.1.1.1. Pressure Inputs and Hydrostatic Head ... 257
7.3.3.1.1.2. Defining Total Pressure and Temperature ... 257
7.3.3.1.1.3. Defining the Flow Direction ... 259
7.3.3.1.1.4. Defining Static Pressure ... 261
7.3.3.1.1.5. Defining Turbulence Parameters ... 262
7.3.3.1.1.6. Defining Radiation Parameters ... 262
7.3.3.1.1.7. Defining Species Mass or Mole Fractions ... 262
7.3.3.1.1.8. Defining Non-Premixed Combustion Parameters ... 262
7.3.3.1.1.9. Defining Premixed Combustion Boundary Conditions ... 262
7.3.3.1.1.10. Defining Discrete Phase Boundary Conditions ... 262

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.3.1.1.11. Defining Multiphase Boundary Conditions 262
7.3.3.1.1.12. Defining Open Channel Boundary Conditions 263
7.3.3.2. Default Settings at Pressure Inlet Boundaries 263
7.3.3.3. Calculation Procedure at Pressure Inlet Boundaries 263
7.3.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries 263
7.3.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries 264
7.3.4. Velocity Inlet Boundary Conditions 264
7.3.4.1. Inputs at Velocity Inlet Boundaries 265
7.3.4.1.1. Summary 265
7.3.4.1.2. Defining the Velocity 266
7.3.4.1.3. Setting the Velocity Magnitude and Direction 267
7.3.4.1.4. Setting the Velocity Magnitude Normal to the Boundary 267
7.3.4.1.5. Setting the Velocity Components 267
7.3.4.1.6. Setting the Angular Velocity 268
7.3.4.1.7. Defining Static Pressure 268
7.3.4.1.8. Defining the Temperature 268
7.3.4.1.9. Defining Outflow Gauge Pressure 268
7.3.4.1.10. Defining Turbulence Parameters 268
7.3.4.1.11. Defining Radiation Parameters 268
7.3.4.1.12. Defining Species Mass or Mole Fractions 269
7.3.4.1.13. Defining Non-Premixed Combustion Parameters 269
7.3.4.1.14. Defining Premixed Combustion Boundary Conditions 269
7.3.4.1.15. Defining Discrete Phase Boundary Conditions 269
7.3.4.1.16. Defining Multiphase Boundary Conditions 269
7.3.4.2. Default Settings at Velocity Inlet Boundaries 269
7.3.4.3. Calculation Procedure at Velocity Inlet Boundaries 269
7.3.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets 270
7.3.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits 270
7.3.4.3.3. Density Calculation 270
7.3.5. Mass Flow Inlet Boundary Conditions 270
7.3.5.1. Limitations and Special Considerations 271
7.3.5.2. Inputs at Mass Flow Inlet Boundaries 271
7.3.5.2.1. Summary 271
7.3.5.2.2. Selecting the Reference Frame 272
7.3.5.2.3. Defining the Mass Flow Rate or Mass Flux 272
7.3.5.2.4. More About Mass Flux and Average Mass Flux 273
7.3.5.2.5. Defining the Total Temperature 273
7.3.5.2.6. Defining Static Pressure 274
7.3.5.2.7. Defining the Flow Direction 274
7.3.5.2.8. Defining Turbulence Parameters 276
7.3.5.2.9. Defining Radiation Parameters 276
7.3.5.2.10. Defining Species Mass or Mole Fractions 276
7.3.5.2.11. Defining Non-Premixed Combustion Parameters 276
7.3.5.2.12. Defining Premixed Combustion Boundary Conditions 276
7.3.5.2.13. Defining Discrete Phase Boundary Conditions 276
7.3.5.2.14. Defining Open Channel Boundary Conditions 276
7.3.5.3. Default Settings at Mass Flow Inlet Boundaries 276
7.3.5.4. Calculation Procedure at Mass Flow Inlet Boundaries 277
7.3.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases 277
7.3.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows 278
7.3.5.4.3. Flux Calculations at Mass Flow Boundaries 278
7.3.6. Inlet Vent Boundary Conditions 278

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.6.1. Inputs at Inlet Vent Boundaries ... 278
7.3.6.1.1. Specifying the Loss Coefficient ... 279
7.3.7. Intake Fan Boundary Conditions ... 280
7.3.7.1. Inputs at Intake Fan Boundaries ... 280
7.3.7.1.1. Specifying the Pressure Jump ... 281
7.3.8. Pressure Outlet Boundary Conditions ... 282
7.3.8.1. Inputs at Pressure Outlet Boundaries ... 282
7.3.8.1.1. Summary ... 282
7.3.8.1.2. Defining Static Pressure ... 283
7.3.8.1.3. Defining Backflow Conditions ... 284
7.3.8.1.4. Defining Radiation Parameters ... 285
7.3.8.1.5. Defining Discrete Phase Boundary Conditions ... 285
7.3.8.1.6. Defining Open Channel Boundary Conditions ... 285
7.3.8.2. Default Settings at Pressure Outlet Boundaries ... 285
7.3.8.3. Calculation Procedure at Pressure Outlet Boundaries ... 285
7.3.8.3.1. Pressure-Based Solver Implementation ... 286
7.3.8.3.2. Density-Based Solver Implementation ... 286
7.3.8.4. Other Optional Inputs at Pressure Outlet Boundaries ... 288
7.3.8.4.1. Non-Reflecting Boundary Conditions Option ... 288
7.3.8.4.2. Target Mass Flow Rate Option ... 288
7.3.8.4.3. Limitations ... 288
7.3.8.4.4. Target Mass Flow Rate Settings ... 288
7.3.8.4.5. Solution Strategies When Using the Target Mass Flow Rate Option ... 290
7.3.8.4.6. Setting Target Mass Flow Rates Using UDFs ... 290
7.3.9. Pressure Far-Field Boundary Conditions ... 290
7.3.9.1. Limitations ... 290
7.3.9.2. Inputs at Pressure Far-Field Boundaries ... 291
7.3.9.2.1. Summary ... 291
7.3.9.2.2. Defining Static Pressure, Mach Number, and Static Temperature ... 292
7.3.9.2.3. Defining the Flow Direction ... 292
7.3.9.2.4. Defining Turbulence Parameters ... 292
7.3.9.2.5. Defining Radiation Parameters ... 292
7.3.9.2.6. Defining Species Transport Parameters ... 292
7.3.9.3. Defining Discrete Phase Boundary Conditions ... 292
7.3.9.4. Default Settings at Pressure Far-Field Boundaries ... 292
7.3.9.5. Calculation Procedure at Pressure Far-Field Boundaries ... 293
7.3.10. Outflow Boundary Conditions ... 293
7.3.10.1. ANSYS FLUENT's Treatment at Outflow Boundaries ... 294
7.3.10.2. Using Outflow Boundaries ... 294
7.3.10.3. Mass Flow Split Boundary Conditions ... 295
7.3.10.4. Other Inputs at Outflow Boundaries ... 296
7.3.10.4.1. Radiation Inputs at Outflow Boundaries ... 296
7.3.10.4.2. Defining Discrete Phase Boundary Conditions ... 297
7.3.11. Outlet Vent Boundary Conditions ... 297
7.3.11.1. Inputs at Outlet Vent Boundaries ... 297
7.3.11.1.1. Specifying the Loss Coefficient ... 298
7.3.12. Exhaust Fan Boundary Conditions ... 299
7.3.12.1. Inputs at Exhaust Fan Boundaries ... 299
7.3.12.1.1. Specifying the Pressure Jump ... 299
7.3.13. Wall Boundary Conditions ... 300
7.3.13.1. Inputs at Wall Boundaries ... 300
7.3.13.1.1. Summary ... 300

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.13.2. Wall Motion ... 301
7.3.13.2.1. Defining a Stationary Wall ... 302
7.3.13.2.2. Velocity Conditions for Moving Walls ... 302
7.3.13.2.3. Shear Conditions at Walls ... 303
7.3.13.2.4. No-Slip Walls ... 304
7.3.13.2.5. Specified Shear ... 304
7.3.13.2.6. Specularity Coefficient ... 304
7.3.13.2.7. Marangoni Stress ... 305
7.3.13.2.8. Wall Roughness Effects in Turbulent Wall-Bounded Flows ... 306
7.3.13.2.9. Law-of-the-Wall Modified for Roughness ... 306
7.3.13.2.10. Setting the Roughness Parameters ... 308
7.3.13.3. Thermal Boundary Conditions at Walls ... 309
7.3.13.3.1. Heat Flux Boundary Conditions ... 310
7.3.13.3.2. Temperature Boundary Conditions ... 310
7.3.13.3.3. Convective Heat Transfer Boundary Conditions ... 310
7.3.13.3.4. External Radiation Boundary Conditions ... 310
7.3.13.3.5. Combined Convection and External Radiation Boundary Conditions ... 310
7.3.13.3.6. Thin-Wall Thermal Resistance Parameters ... 311
7.3.13.3.7. Thermal Conditions for Two-Sided Walls ... 312
7.3.13.3.8. Shell Conduction in Thin-Walls ... 313
7.3.13.4. Species Boundary Conditions for Walls ... 314
7.3.13.4.1. Reaction Boundary Conditions for Walls ... 315
7.3.13.5. Radiation Boundary Conditions for Walls ... 315
7.3.13.6. Discrete Phase Model (DPM) Boundary Conditions for Walls ... 316
7.3.13.6.1. Wall Adhesion Contact Angle for VOF Model ... 316
7.3.13.7. User-Defined Scalar (UDS) Boundary Conditions for Walls ... 316
7.3.13.8. Default Settings at Wall Boundaries ... 316
7.3.13.9. Shear-Stress Calculation Procedure at Wall Boundaries ... 316
7.3.13.9.1. Shear-Stress Calculation in Laminar Flow ... 316
7.3.13.9.2. Shear-Stress Calculation in Turbulent Flows ... 317
7.3.13.10. Heat Transfer Calculations at Wall Boundaries ... 317
7.3.13.10.1. Temperature Boundary Conditions ... 317
7.3.13.10.2. Heat Flux Boundary Conditions ... 317
7.3.13.10.3. Convective Heat Transfer Boundary Conditions ... 318
7.3.13.10.4. External Radiation Boundary Conditions ... 318
7.3.13.10.5. Combined External Convection and Radiation Boundary Conditions ... 318
7.3.13.10.6. Calculation of the Fluid-Side Heat Transfer Coefficient ... 319
7.3.14. Symmetry Boundary Conditions ... 319
7.3.14.1. Examples of Symmetry Boundaries ... 319
7.3.14.2. Calculation Procedure at Symmetry Boundaries ... 321
7.3.15. Periodic Boundary Conditions ... 321
7.3.15.1. Examples of Periodic Boundaries ... 321
7.3.15.2. Inputs for Periodic Boundaries ... 321
7.3.15.3. Default Settings at Periodic Boundaries ... 323
7.3.15.4. Calculation Procedure at Periodic Boundaries ... 323
7.3.16. Axis Boundary Conditions ... 323
7.3.16.1. Calculation Procedure at Axis Boundaries ... 323
7.3.17. Fan Boundary Conditions ... 324
7.3.17.1. Fan Equations ... 324
7.3.17.1.1. Modeling the Pressure Rise Across the Fan ... 324
7.3.17.1.2. Modeling the Fan Swirl Velocity ... 324
7.3.17.2. User Inputs for Fans ... 325

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.17.2.1. Identifying the Fan Zone ... 325
7.3.17.2.2. Defining the Pressure Jump ... 326
7.3.17.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function ... 326
7.3.17.2.2.2. Constant Value ... 327
7.3.17.2.2.3. User-Defined Function or Profile ... 327
7.3.17.2.2.4. Example: Determining the Pressure Jump Function ... 327
7.3.17.2.3. Defining Discrete Phase Boundary Conditions for the Fan ... 328
7.3.17.2.4. Defining the Fan Swirl Velocity ... 328
7.3.17.2.4.1. Polynomial Function ... 329
7.3.17.2.4.2. Constant Value ... 329
7.3.17.2.4.3. User-Defined Function or Profile ... 329
7.3.17.3. Postprocessing for Fans ... 330
7.3.17.3.1. Reporting the Pressure Rise Through the Fan ... 330
7.3.17.3.2. Graphical Plots ... 330
7.3.18. Radiator Boundary Conditions ... 330
7.3.18.1. Radiator Equations ... 330
7.3.18.1.1. Modeling the Pressure Loss Through a Radiator ... 330
7.3.18.1.2. Modeling the Heat Transfer Through a Radiator ... 331
7.3.18.1.2.1. Calculating the Heat Transfer Coefficient ... 331
7.3.18.2. User Inputs for Radiators ... 332
7.3.18.2.1. Identifying the Radiator Zone ... 333
7.3.18.2.2. Defining the Pressure Loss Coefficient Function ... 333
7.3.18.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function ... 333
7.3.18.2.2.2. Constant Value ... 334
7.3.18.2.2.3. Example: Calculating the Loss Coefficient ... 334
7.3.18.2.3. Defining the Heat Flux Parameters ... 335
7.3.18.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function ... 335
7.3.18.2.3.2. Constant Value ... 336
7.3.18.2.3.3. Example: Determining the Heat Transfer Coefficient Function ... 336
7.3.18.2.4. Defining Discrete Phase Boundary Conditions for the Radiator ... 336
7.3.18.3. Postprocessing for Radiators ... 336
7.3.18.3.1. Reporting the Radiator Pressure Drop ... 336
7.3.18.3.2. Reporting Heat Transfer in the Radiator ... 337
7.3.18.3.3. Graphical Plots ... 337
7.3.19. Porous Jump Boundary Conditions ... 337
7.3.19.1. User Inputs for the Porous Jump Model ... 338
7.3.19.1.1. Identifying the Porous Jump Zone ... 338
7.3.19.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump ... 339
7.3.19.2. Postprocessing for the Porous Jump ... 339
7.4. Non-Reflecting Boundary Conditions ... 339
7.4.1. Turbo-Specific Non-Reflecting Boundary Conditions ... 339
7.4.1.1. Overview ... 339
7.4.1.2. Limitations ... 340
7.4.1.3. Theory ... 342
7.4.1.3.1. Equations in Characteristic Variable Form ... 342
7.4.1.3.2. Inlet Boundary ... 344
7.4.1.3.3. Outlet Boundary ... 348
7.4.1.3.4. Updated Flow Variables ... 349
7.4.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions ... 350
7.4.1.4.1. Using the NRBCs with the Mixing-Plane Model ... 351
7.4.1.4.2. Using the NRBCs in Parallel ANSYS FLUENT ... 351
7.4.2. General Non-Reflecting Boundary Conditions ... 351

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.2.1. Overview ... 351
7.4.2.2. Restrictions and Limitations ... 352
7.4.2.3. Theory ... 352
7.4.2.4. Using General Non-Reflecting Boundary Conditions ... 357
7.5. User-Defined Fan Model ... 359
7.5.1. Steps for Using the User-Defined Fan Model ... 359
7.5.2. Example of a User-Defined Fan ... 359
7.5.2.1. Setting the User-Defined Fan Parameters ... 360
7.5.2.2. Sample User-Defined Fan Program ... 361
7.5.2.3. Initializing the Flow Field and Profile Files ... 363
7.5.2.4. Selecting the Profiles ... 363
7.5.2.5. Performing the Calculation ... 364
7.5.2.6. Results ... 365
7.6. Profiles ... 366
7.6.1. Profile Specification Types ... 366
7.6.2. Profile File Format ... 367
7.6.2.1. Example ... 368
7.6.3. Using Profiles ... 368
7.6.3.1. Checking and Deleting Profiles ... 369
7.6.3.2. Viewing Profile Data ... 370
7.6.3.3. Example ... 370
7.6.4. Reorienting Profiles ... 371
7.6.4.1. Steps for Changing the Profile Orientation ... 371
7.6.4.2. Profile Orienting Example ... 374
7.6.5. Defining Transient Cell Zone and Boundary Conditions ... 376
7.6.5.1. Standard Transient Profiles ... 377
7.6.5.2. Tabular Transient Profiles ... 378
7.7. Coupling Boundary Conditions with GT-Power ... 379
7.7.1. Requirements and Restrictions ... 379
7.7.2. User Inputs ... 380
7.8. Coupling Boundary Conditions with WAVE ... 381
7.8.1. Requirements and Restrictions ... 381
7.8.2. User Inputs ... 382
8. Physical Properties ... 385
8.1. Defining Materials ... 385
8.1.1. Physical Properties for Solid Materials ... 386
8.1.2. Material Types and Databases ... 386
8.1.3. Using the Materials Task Page ... 387
8.1.3.1. Modifying Properties of an Existing Material ... 388
8.1.3.2. Renaming an Existing Material ... 388
8.1.3.3. Copying Materials from the ANSYS FLUENT Database ... 389
8.1.3.4. Creating a New Material ... 390
8.1.3.5. Saving Materials and Properties ... 390
8.1.3.6. Deleting a Material ... 391
8.1.3.7. Changing the Order of the Materials List ... 391
8.1.4. Using a User-Defined Materials Database ... 391
8.1.4.1. Opening a User-Defined Database ... 392
8.1.4.2. Viewing Materials in a User-Defined Database ... 392
8.1.4.3. Copying Materials from a User-Defined Database ... 393
8.1.4.4. Copying Materials from the Case to a User-Defined Database ... 394
8.1.4.5. Modifying Properties of an Existing Material ... 395
8.1.4.6. Creating a New Materials Database and Materials ... 395

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.4.7. Deleting Materials from a Database ... 398
8.2. Defining Properties Using Temperature-Dependent Functions ... 398
8.2.1. Inputs for Polynomial Functions ... 399
8.2.2. Inputs for Piecewise-Linear Functions ... 400
8.2.3. Inputs for Piecewise-Polynomial Functions ... 401
8.2.4. Checking and Modifying Existing Profiles ... 402
8.3. Density ... 403
8.3.1. Defining Density for Various Flow Regimes ... 403
8.3.1.1. Mixing Density Relationships in Multiple-Zone Models ... 403
8.3.2. Input of Constant Density ... 404
8.3.3. Inputs for the Boussinesq Approximation ... 404
8.3.4. Density as a Profile Function of Temperature ... 404
8.3.5. Incompressible Ideal Gas Law ... 405
8.3.5.1. Density Inputs for the Incompressible Ideal Gas Law ... 405
8.3.6. Ideal Gas Law for Compressible Flows ... 406
8.3.6.1. Density Inputs for the Ideal Gas Law for Compressible Flows ... 406
8.3.7. Composition-Dependent Density for Multicomponent Mixtures ... 407
8.4. Viscosity ... 408
8.4.1. Input of Constant Viscosity ... 409
8.4.2. Viscosity as a Function of Temperature ... 409
8.4.2.1. Sutherland Viscosity Law ... 410
8.4.2.1.1. Inputs for Sutherland's Law ... 410
8.4.2.2. Power-Law Viscosity Law ... 411
8.4.2.2.1. Inputs for the Power Law ... 412
8.4.3. Defining the Viscosity Using Kinetic Theory ... 412
8.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures ... 412
8.4.5. Viscosity for Non-Newtonian Fluids ... 413
8.4.5.1. Temperature Dependent Viscosity ... 414
8.4.5.2. Power Law for Non-Newtonian Viscosity ... 415
8.4.5.2.1. Inputs for the Non-Newtonian Power Law ... 415
8.4.5.3. The Carreau Model for Pseudo-Plastics ... 415
8.4.5.3.1. Inputs for the Carreau Model ... 416
8.4.5.4. Cross Model ... 416
8.4.5.4.1. Inputs for the Cross Model ... 417
8.4.5.5. Herschel-Bulkley Model for Bingham Plastics ... 417
8.4.5.5.1. Inputs for the Herschel-Bulkley Model ... 418
8.5. Thermal Conductivity ... 418
8.5.1. Constant Thermal Conductivity ... 419
8.5.2. Thermal Conductivity as a Function of Temperature ... 419
8.5.3. Thermal Conductivity Using Kinetic Theory ... 420
8.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures ... 420
8.5.5. Anisotropic Thermal Conductivity for Solids ... 422
8.5.5.1. Anisotropic Thermal Conductivity ... 422
8.5.5.2. Biaxial Thermal Conductivity ... 423
8.5.5.3. Orthotropic Thermal Conductivity ... 424
8.5.5.4. Cylindrical Orthotropic Thermal Conductivity ... 426
8.6. User-Defined Scalar (UDS) Diffusivity ... 427
8.6.1. Isotropic Diffusion ... 427
8.6.2. Anisotropic Diffusion ... 428
8.6.2.1. Anisotropic Diffusivity ... 429
8.6.2.2. Orthotropic Diffusivity ... 430
8.6.2.3. Cylindrical Orthotropic Diffusivity ... 431

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.3. User-Defined Anisotropic Diffusivity ... 432
8.7. Specific Heat Capacity ... 433
8.7.1. Input of Constant Specific Heat Capacity ... 434
8.7.2. Specific Heat Capacity as a Function of Temperature ... 434
8.7.3. Defining Specific Heat Capacity Using Kinetic Theory ... 434
8.7.4. Specific Heat Capacity as a Function of Composition ... 435
8.8. Radiation Properties ... 435
8.8.1. Absorption Coefficient ... 436
8.8.1.1. Inputs for a Constant Absorption Coefficient ... 436
8.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient ... 436
8.8.1.2.1. Path Length Inputs ... 436
8.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient ... 437
8.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient ... 437
8.8.2. Scattering Coefficient ... 437
8.8.2.1. Inputs for a Constant Scattering Coefficient ... 437
8.8.2.2. Inputs for the Scattering Phase Function ... 437
8.8.2.2.1. Isotropic Phase Function ... 438
8.8.2.2.2. Linear-Anisotropic Phase Function ... 438
8.8.2.2.3. Delta-Eddington Phase Function ... 438
8.8.2.2.4. User-Defined Phase Function ... 438
8.8.3. Refractive Index ... 438
8.8.4. Reporting the Radiation Properties ... 438
8.9. Mass Diffusion Coefficients ... 438
8.9.1. Fickian Diffusion ... 439
8.9.2. Full Multicomponent Diffusion ... 440
8.9.2.1. General Theory ... 440
8.9.2.2. Maxwell-Stefan Equations ... 440
8.9.3. Thermal Diffusion Coefficients ... 442
8.9.4. Mass Diffusion Coefficient Inputs ... 442
8.9.4.1. Constant Dilute Approximation Inputs ... 443
8.9.4.2. Dilute Approximation Inputs ... 443
8.9.4.3. Multicomponent Method Inputs ... 444
8.9.4.4. Thermal Diffusion Coefficient Inputs ... 445
8.9.5. Mass Diffusion Coefficient Inputs for Turbulent Flow ... 446
8.10. Standard State Enthalpies ... 447
8.11. Standard State Entropies ... 447
8.12. Molecular Heat Transfer Coefficient ... 448
8.13. Kinetic Theory Parameters ... 448
8.13.1. Inputs for Kinetic Theory ... 448
8.14. Operating Pressure ... 449
8.14.1. The Effect of Numerical Roundoff on Pressure Calculation in Low-Mach-Number Flow ... 449
8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure ... 449
8.14.3. Setting the Operating Pressure ... 449
8.14.3.1. The Significance of Operating Pressure ... 449
8.14.3.2. How to Set the Operating Pressure ... 450
8.15. Reference Pressure Location ... 450
8.15.1. Actual Reference Pressure Location ... 451
8.16. Real Gas Models ... 451
8.16.1. Introduction ... 452
8.16.2. Choosing a Real Gas Model ... 453
8.16.3. Cubic Equation of State Models ... 454
8.16.3.1. Overview and Limitations ... 454

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.3.2. Equation of State ... 456
8.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations ... 458
8.16.3.4. Critical Constants for Pure Components ... 460
8.16.3.5. Calculations for Mixtures ... 460
8.16.3.5.1. Using the Cubic Equation of State Real Gas Models ... 462
8.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models ... 465
8.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models ... 465
8.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model ... 467
8.16.4. The NIST Real Gas Models ... 468
8.16.4.1. Limitations of the NIST Real Gas Models ... 468
8.16.4.2. The REFPROP v7.0 Database ... 468
8.16.4.3. Using the NIST Real Gas Models ... 469
8.16.4.3.1. Activating the NIST Real Gas Model ... 470
8.16.4.4. Solution Strategies and Considerations for NIST Real Gas Model Simulation ... 471
8.16.4.4.1. Writing Your Case File ... 472
8.16.4.4.2. Postprocessing ... 472
8.16.5. The User-Defined Real Gas Model ... 472
8.16.5.1. Limitations of the User-Defined Real Gas Model ... 473
8.16.5.2. Writing the UDRGM C Function Library ... 475
8.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File ... 478
8.16.5.3.1. Compiling the UDRGM Using the Graphical Interface ... 478
8.16.5.3.2. Compiling the UDRGM Using the Text Interface ... 478
8.16.5.3.3. Loading the UDRGM Shared Library File ... 479
8.16.5.4. UDRGM Example: Ideal Gas Equation of State ... 480
8.16.5.4.1. Ideal Gas UDRGM Code Listing ... 481
8.16.5.5. Additional UDRGM Examples ... 483
9. Modeling Basic Fluid Flow ... 485
9.1. User-Defined Scalar (UDS) Transport Equations ... 485
9.1.1. Introduction ... 485
9.1.2. UDS Theory ... 485
9.1.2.1. Single Phase Flow ... 485
9.1.2.2. Multiphase Flow ... 486
9.1.3. Setting Up UDS Equations in ANSYS FLUENT ... 488
9.1.3.1. Single Phase Flow ... 488
9.1.3.2. Multiphase Flow ... 492
9.2. Periodic Flows ... 493
9.2.1. Overview and Limitations ... 493
9.2.1.1. Overview ... 493
9.2.1.2. Limitations for Modeling Streamwise-Periodic Flow ... 494
9.2.2. User Inputs for the Pressure-Based Solver ... 494
9.2.2.1. Setting Parameters for the Calculation of β ... 496
9.2.3. User Inputs for the Density-Based Solvers ... 496
9.2.4. Monitoring the Value of the Pressure Gradient ... 497
9.2.5. Postprocessing for Streamwise-Periodic Flows ... 497
9.3. Swirling and Rotating Flows ... 497
9.3.1. Overview of Swirling and Rotating Flows ... 498
9.3.1.1. Axisymmetric Flows with Swirl or Rotation ... 498
9.3.1.1.1. Momentum Conservation Equation for Swirl Velocity ... 498
9.3.1.2. Three-Dimensional Swirling Flows ... 498
9.3.1.3. Flows Requiring a Moving Reference Frame ... 499

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.2. Turbulence Modeling in Swirling Flows ... 499
9.3.3. Mesh Setup for Swirling and Rotating Flows ... 500
9.3.3.1. Coordinate System Restrictions ... 500
9.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows ... 500
9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation ... 500
9.3.4.1. Problem Setup for Axisymmetric Swirling Flows ... 501
9.3.4.2. Solution Strategies for Axisymmetric Swirling Flows ... 501
9.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows ... 502
9.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed ... 503
9.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows ... 503
9.4. Compressible Flows ... 503
9.4.1. When to Use the Compressible Flow Model ... 505
9.4.2. Physics of Compressible Flows ... 505
9.4.2.1. Basic Equations for Compressible Flows ... 506
9.4.2.2. The Compressible Form of the Gas Law ... 506
9.4.3. Modeling Inputs for Compressible Flows ... 506
9.4.3.1. Boundary Conditions for Compressible Flows ... 507
9.4.4. Floating Operating Pressure ... 507
9.4.4.1. Limitations ... 508
9.4.4.2. Theory ... 508
9.4.4.3. Enabling Floating Operating Pressure ... 508
9.4.4.4. Setting the Initial Value for the Floating Operating Pressure ... 509
9.4.4.5. Storage and Reporting of the Floating Operating Pressure ... 509
9.4.4.6. Monitoring Absolute Pressure ... 509
9.4.5. Solution Strategies for Compressible Flows ... 509
9.4.6. Reporting of Results for Compressible Flows ... 510
9.5. Inviscid Flows ... 510
9.5.1. Setting Up an Inviscid Flow Model ... 510
9.5.2. Solution Strategies for Inviscid Flows ... 511
9.5.3. Postprocessing for Inviscid Flows ... 511
10. Modeling Flows with Moving Reference Frames ... 513
10.1. Introduction ... 513
10.2. Flow in Single Moving Reference Frames (SRF) ... 514
10.2.1. Mesh Setup for a Single Moving Reference Frame ... 515
10.2.2. Setting Up a Single Moving Reference Frame Problem ... 516
10.2.2.1. Choosing the Relative or Absolute Velocity Formulation ... 518
10.2.2.1.1. Example ... 518
10.2.3. Solution Strategies for a Single Moving Reference Frame ... 519
10.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability ... 519
10.2.4. Postprocessing for a Single Moving Reference Frame ... 520
10.3. Flow in Multiple Moving Reference Frames ... 521
10.3.1. The Multiple Reference Frame Model ... 522
10.3.1.1. Overview ... 522
10.3.1.2. Limitations ... 522
10.3.2. The Mixing Plane Model ... 523
10.3.2.1. Overview ... 523
10.3.2.2. Limitations ... 524
10.3.3. Mesh Setup for a Multiple Moving Reference Frame ... 524
10.3.4. Setting Up a Multiple Moving Reference Frame Problem ... 524
10.3.4.1. Setting Up Multiple Reference Frames ... 524
10.3.4.2. Setting Up the Mixing Plane Model ... 528

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.4.2.1. Modeling Options ... 531
10.3.4.2.1.1. Fixing the Pressure Level for an Incompressible Flow ... 531
10.3.4.2.1.2. Conserving Swirl Across the Mixing Plane ... 532
10.3.4.2.1.3. Conserving Total Enthalpy Across the Mixing Plane ... 532
10.3.5. Solution Strategies for MRF and Mixing Plane Problems ... 532
10.3.5.1. MRF Model ... 532
10.3.5.2. Mixing Plane Model ... 532
10.3.6. Postprocessing for MRF and Mixing Plane Problems ... 533
11. Modeling Flows Using Sliding and Dynamic Meshes ... 535
11.1. Introduction ... 535
11.2. Sliding Mesh Examples ... 535
11.3. The Sliding Mesh Technique ... 537
11.4. Sliding Mesh Interface Shapes ... 538
11.5. Using Sliding Meshes ... 540
11.5.1. Requirements and Constraints ... 540
11.5.2. Setting Up the Sliding Mesh Problem ... 541
11.5.3. Solution Strategies for Sliding Meshes ... 545
11.5.3.1. Saving Case and Data Files ... 545
11.5.3.2. Time-Periodic Solutions ... 546
11.5.4. Postprocessing for Sliding Meshes ... 547
11.6. Using Dynamic Meshes ... 548
11.6.1. Setting Dynamic Mesh Modeling Parameters ... 550
11.6.2. Dynamic Mesh Update Methods ... 551
11.6.2.1. Smoothing Methods ... 551
11.6.2.1.1. Spring-Based Smoothing ... 552
11.6.2.1.1.1. Applicability of the Spring-Based Smoothing Methods ... 555
11.6.2.1.2. Diffusion-Based Smoothing ... 556
11.6.2.1.2.1. Applicability of Diffusion-Based Smoothing Method ... 560
11.6.2.1.3. Laplacian Smoothing Method ... 560
11.6.2.1.4. Boundary Layer Smoothing Method ... 561
11.6.2.2. Dynamic Layering Method ... 564
11.6.2.2.1. Applicability of the Dynamic Layering Method ... 567
11.6.2.3. Remeshing Methods ... 567
11.6.2.3.1. Local Remeshing Method ... 570
11.6.2.3.1.1. Local Cell Remeshing Method ... 570
11.6.2.3.1.1.1. Cell Zone Remeshing Method ... 571
11.6.2.3.1.1.2. Applicability of the Cell Zone Remeshing Method ... 571
11.6.2.3.1.2. Local Face Remeshing Method ... 571
11.6.2.3.1.2.1. Applicability of the Local Face Remeshing Method ... 571
11.6.2.3.1.3. Local Remeshing Based on Size Functions ... 571
11.6.2.3.2. Face Region Remeshing Method ... 577
11.6.2.3.2.1. Applicability of the Face Region Remeshing Method ... 578
11.6.2.3.3. CutCell Zone Remeshing Method ... 579
11.6.2.3.3.1. Applicability of the CutCell Zone Remeshing Method ... 580
11.6.2.3.3.2. Using the CutCell Zone Remeshing Method ... 581
11.6.2.3.3.3. Applying the CutCell Zone Remeshing Method Manually ... 581
11.6.2.3.4. 2.5D Surface Remeshing Method ... 582
11.6.2.3.4.1. Applicability of the 2.5D Surface Remeshing Method ... 583
11.6.2.3.4.2. Using the 2.5D Model ... 583
11.6.2.3.5. Feature Detection ... 586
11.6.2.3.5.1. Applicability of Feature Detection ... 586
11.6.2.4. Volume Mesh Update Procedure ... 586

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.3. In-Cylinder Settings ... 587
11.6.3.1. Using the In-Cylinder Option ... 591
11.6.3.1.1. Overview ... 591
11.6.3.1.2. Defining the Mesh Topology ... 592
11.6.3.1.3. Defining Motion/Geometry Attributes of Mesh Zones ... 594
11.6.3.1.4. Defining Valve Opening and Closure ... 601
11.6.4. Six DOF Solver Settings ... 601
11.6.4.1. Using the Six DOF Solver ... 602
11.6.4.1.1. Setting Rigid Body Motion Attributes for the Six DOF Solver ... 602
11.6.5. Implicit Update Settings ... 603
11.6.6. Defining Dynamic Mesh Events ... 605
11.6.6.1. Procedure for Defining Events ... 605
11.6.6.2. Defining Events for In-Cylinder Applications ... 607
11.6.6.2.1. Events ... 608
11.6.6.2.2. Changing the Zone Type ... 608
11.6.6.2.3. Copying Zone Boundary Conditions ... 608
11.6.6.2.4. Activating a Cell Zone ... 608
11.6.6.2.5. Deactivating a Cell Zone ... 608
11.6.6.2.6. Creating a Sliding Interface ... 608
11.6.6.2.7. Deleting a Sliding Interface ... 609
11.6.6.2.8. Changing the Motion Attribute of a Dynamic Zone ... 610
11.6.6.2.9. Changing the Time Step ... 610
11.6.6.2.10. Changing the Under-Relaxation Factor ... 610
11.6.6.2.11. Inserting a Boundary Zone Layer ... 610
11.6.6.2.12. Removing a Boundary Zone Layer ... 611
11.6.6.2.13. Inserting an Interior Zone Layer ... 611
11.6.6.2.14. Removing an Interior Zone Layer ... 612
11.6.6.2.15. Inserting a Cell Layer ... 612
11.6.6.2.16. Removing a Cell Layer ... 612
11.6.6.2.17. Executing a Command ... 613
11.6.6.2.18. Replacing the Mesh ... 613
11.6.6.3. Exporting and Importing Events ... 613
11.6.7. Specifying the Motion of Dynamic Zones ... 613
11.6.7.1. General Procedure ... 613
11.6.7.1.1. Creating a Dynamic Zone ... 613
11.6.7.1.2. Modifying a Dynamic Zone ... 614
11.6.7.1.3. Checking the Center of Gravity ... 614
11.6.7.1.4. Deleting a Dynamic Zone ... 614
11.6.7.2. Stationary Zones ... 614
11.6.7.3. Rigid Body Motion ... 616
11.6.7.4. Deforming Motion ... 619
11.6.7.5. User-Defined Motion ... 622
11.6.7.5.1. Specifying Boundary Layer Deformation Smoothing ... 623
11.6.7.6. Solid-Body Kinematics ... 623
11.6.8. Previewing the Dynamic Mesh ... 626
11.6.8.1. Previewing Zone Motion ... 626
11.6.8.2. Previewing Mesh Motion ... 627
11.6.9. Steady-State Dynamic Mesh Applications ... 628
11.6.9.1. An Example of Steady-State Dynamic Mesh Usage ... 629
12. Modeling Flows Using the Mesh Morpher/Optimizer ... 631
12.1. Introduction ... 631
12.1.1. Limitations ... 631

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.2. The Optimization Process ... 631
12.3. Optimizers ... 632
12.3.1. The Compass Optimizer ... 632
12.3.2. The Simplex Optimizer ... 633
12.3.3. The Torczon Optimizer ... 633
12.3.4. The Powell Optimizer ... 633
12.3.5. The Rosenbrock Optimizer ... 634
12.4. Setting Up the Mesh Morpher/Optimizer ... 634
13. Modeling Turbulence ... 643
13.1. Introduction ... 643
13.2. Choosing a Turbulence Model ... 644
13.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models ... 645
13.2.1.1. Spalart-Allmaras One-Equation Model ... 645
13.2.1.2. k-ε Models ... 645
13.2.1.3. k-ω Models ... 646
13.2.1.4. RSM Models ... 646
13.2.1.5. Laminar-Turbulent Transition Models ... 646
13.2.1.6. Model Enhancements ... 647
13.2.1.7. Wall Treatment RANS Models ... 647
13.2.1.8. Grid Resolution RANS Models ... 647
13.2.2. Scale-Resolving Simulation (SRS) Models ... 648
13.2.2.1. Large Eddy Simulation (LES) ... 648
13.2.2.2. Hybrid RANS-LES Models ... 649
13.2.2.2.1. Scale-Adaptive Simulation (SAS) ... 649
13.2.2.3. Detached Eddy Simulation (DES) ... 650
13.2.2.4. Zonal Modeling and Embedded LES (E-LES) ... 651
13.2.3. Grid Resolution SRS Models ... 651
13.2.3.1. Wall Boundary Layers ... 651
13.2.3.2. Free Shear Flows ... 651
13.2.4. Numerics Settings for SRS Models ... 652
13.2.4.1. Time Discretization ... 652
13.2.4.2. Spatial Discretization ... 653
13.2.4.3. Iterative Scheme ... 653
13.2.4.3.1. Convergence Control ... 654
13.2.5. Model Hierarchy ... 654
13.3. Steps in Using a Turbulence Model ... 655
13.4. Setting Up the Spalart-Allmaras Model ... 657
13.5. Setting Up the k- ε Model ... 658
13.5.1. Setting Up the Standard or Realizable k- ε Model ... 658
13.5.2. Setting Up the RNG k- ε Model ... 659
13.6. Setting Up the k- ω Model ... 661
13.6.1. Setting Up the Standard k- ω Model ... 661
13.6.2. Setting Up the Shear-Stress Transport k- ω Model ... 662
13.7. Setting Up the Transition k- kl- ω Model ... 662
13.8. Setting Up the Transition SST Model ... 662
13.9. Setting Up the Reynolds Stress Model ... 663
13.10. Setting Up the Scale-Adaptive Simulation (SAS) Model ... 666
13.11. Setting Up the Detached Eddy Simulation Model ... 666
13.11.1. Setting Up the Spalart-Allmaras DES Model ... 667
13.11.2. Setting Up the Realizable k- ε DES Model ... 668
13.11.3. Setting Up the SST k- ω DES Model ... 668
13.12. Setting Up the Large Eddy Simulation Model ... 669

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.13. Setting Up the Embedded Large Eddy Simulation (E-LES) Model 670
13.14. Setup Options for all Turbulence Modeling 673
13.14.1. Including the Viscous Heating Effects 674
13.14.2. Including Turbulence Generation Due to Buoyancy 674
13.14.3. Vorticity- and Strain/Vorticity-Based Production 674
13.14.4. Delayed Detached Eddy Simulation (DDES) 674
13.14.5. Differential Viscosity Modification 674
13.14.6. Swirl Modification 674
13.14.7. Low-Re Corrections 675
13.14.8. Low-Re Damping 675
13.14.9. Shear Flow Corrections 675
13.14.10. Turbulence Damping 675
13.14.11. Including Pressure Gradient Effects 675
13.14.12. Including Thermal Effects 676
13.14.13. Including the Wall Reflection Term 676
13.14.14. Solving the k Equation to Obtain Wall Boundary Conditions 676
13.14.15. Quadratic Pressure-Strain Model 676
13.14.16. Low-Re Stress-Omega Pressure-Strain 677
13.14.17. Subgrid-Scale Model 677
13.14.18. Customizing the Turbulent Viscosity 677
13.14.19. Customizing the Turbulent Prandtl and Schmidt Numbers 677
13.14.20. Modeling Turbulence with Non-Newtonian Fluids 678
13.14.21. Shielding Functions for the SST Detached Eddy Simulation Model 678
13.15. Defining Turbulence Boundary Conditions 678
13.15.1. The Spalart-Allmaras Model 678
13.15.2. k-ε Models and k-ω Models 678
13.15.3. Reynolds Stress Model 679
13.15.4. Large Eddy Simulation Model 680
13.16. Providing an Initial Guess for k and ε (or k and ω) 681
13.17. Solution Strategies for Turbulent Flow Simulations 681
13.17.1. Mesh Generation 682
13.17.2. Accuracy 682
13.17.3. Convergence 682
13.17.4. RSM-Specific Solution Strategies 683
13.17.4.1. Under-Relaxation of the Reynolds Stresses 683
13.17.4.2. Disabling Calculation Updates of the Reynolds Stresses 683
13.17.4.3. Residual Reporting for the RSM 683
13.17.5. LES-Specific Solution Strategies 684
13.17.5.1. Temporal Discretization 684
13.17.5.2. Spatial Discretization 684
13.18. Postprocessing for Turbulent Flows 685
13.18.1. Custom Field Functions for Turbulence 689
13.18.2. Postprocessing Turbulent Flow Statistics 689
13.18.3. Troubleshooting 690
14. Modeling Heat Transfer 691
14.1. Introduction 691
14.2. Modeling Conductive and Convective Heat Transfer 691
14.2.1. Solving Heat Transfer Problems 691
14.2.1.1. Limiting the Predicted Temperature Range 693
14.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions 693
14.2.2. Solution Strategies for Heat Transfer Modeling 693
14.2.2.1. Under-Relaxation of the Energy Equation 693

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved 694
14.2.2.3. Disabling the Species Diffusion Term 694
14.2.2.4. Step-by-Step Solutions 694
14.2.2.4.1. Decoupled Flow and Heat Transfer Calculations 694
14.2.2.4.2. Coupled Flow and Heat Transfer Calculations 695
14.2.3. Postprocessing Heat Transfer Quantities 695
14.2.3.1. Available Variables for Postprocessing 695
14.2.3.2. Definition of Enthalpy and Energy in Reports and Displays 696
14.2.3.3. Reporting Heat Transfer Through Boundaries 696
14.2.3.4. Reporting Heat Transfer Through a Surface 696
14.2.3.5. Reporting Averaged Heat Transfer Coefficients 696
14.2.3.6. Exporting Heat Flux Data 696
14.2.4. Natural Convection and Buoyancy-Driven Flows 697
14.2.4.1. Modeling Natural Convection in a Closed Domain 697
14.2.4.2. The Boussinesq Model 697
14.2.4.3. Limitations of the Boussinesq Model 698
14.2.4.4. Steps in Solving Buoyancy-Driven Flow Problems 698
14.2.4.5. Operating Density 700
14.2.4.5.1. Setting the Operating Density 700
14.2.4.6. Solution Strategies for Buoyancy-Driven Flows 700
14.2.4.6.1. Guidelines for Solving High-Rayleigh-Number Flows 700
14.2.4.7. Postprocessing Buoyancy-Driven Flows 701
14.2.5. Shell Conduction Considerations 701
14.2.5.1. Introduction 701
14.2.5.2. Physical Treatment 702
14.2.5.3. Limitations of Shell Conduction Walls 702
14.2.5.4. Managing Shell Conduction Walls 703
14.2.5.5. Initialization 704
14.2.5.6. Postprocessing 704
14.3. Modeling Radiation 704
14.3.1. Using the Radiation Models 705
14.3.2. Setting Up the P-1 Model with Non-Gray Radiation 706
14.3.3. Setting Up the DTRM 707
14.3.3.1. Defining the Rays 707
14.3.3.2. Controlling the Clusters 708
14.3.3.3. Controlling the Rays 708
14.3.3.4. Writing and Reading the DTRM Ray File 709
14.3.3.5. Displaying the Clusters 709
14.3.4. Setting Up the S2S Model 709
14.3.4.1. View Factors and Clustering Settings 711
14.3.4.1.1. Forming Surface Clusters 711
14.3.4.1.1.1. Setting the Split Angle for Clusters 713
14.3.4.1.2. Setting Up the View Factor Calculation 713
14.3.4.1.2.1. Selecting the Basis for Computing View Factors 713
14.3.4.1.2.2. Selecting the Method for Computing View Factors 714
14.3.4.1.2.3. Accounting for Blocking Surfaces 715
14.3.4.1.2.4. Specifying Boundary Zone Participation 715
14.3.4.2. Computing View Factors 717
14.3.4.2.1. Computing View Factors Inside ANSYS FLUENT 717
14.3.4.2.2. Computing View Factors Outside ANSYS FLUENT 718
14.3.4.3. Reading View Factors into ANSYS FLUENT 720
14.3.5. Setting Up the DO Model 720

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.5.1. Angular Discretization ... 720
14.3.5.2. Defining Non-Gray Radiation for the DO Model ... 721
14.3.5.3. Enabling DO/Energy Coupling ... 723
14.3.6. Defining Material Properties for Radiation ... 723
14.3.6.1. Absorption Coefficient for a Non-Gray Model ... 724
14.3.6.2. Refractive Index for a Non-Gray Model ... 724
14.3.7. Defining Boundary Conditions for Radiation ... 724
14.3.7.1. Inlet and Exit Boundary Conditions ... 724
14.3.7.1.1. Emissivity ... 724
14.3.7.1.2. Black Body Temperature ... 724
14.3.7.2. Wall Boundary Conditions for the DTRM, and the P-1, S2S, and Rosseland Models ... 725
14.3.7.2.1. Boundary Conditions for the S2S Model ... 725
14.3.7.3. Wall Boundary Conditions for the DO Model ... 726
14.3.7.3.1. Opaque Walls ... 726
14.3.7.3.2. Semi-Transparent Walls ... 728
14.3.7.4. Solid Cell Zones Conditions for the DO Model ... 731
14.3.7.5. Thermal Boundary Conditions ... 732
14.3.8. Solution Strategies for Radiation Modeling ... 732
14.3.8.1. P-1 Model Solution Parameters ... 733
14.3.8.2. DTRM Solution Parameters ... 733
14.3.8.3. S2S Solution Parameters ... 734
14.3.8.4. DO Solution Parameters ... 735
14.3.8.5. Running the Calculation ... 735
14.3.8.5.1. Residual Reporting for the P-1 Model ... 735
14.3.8.5.2. Residual Reporting for the DO Model ... 735
14.3.8.5.3. Residual Reporting for the DTRM ... 735
14.3.8.5.4. Residual Reporting for the S2S Model ... 736
14.3.8.5.5. Disabling the Update of the Radiation Fluxes ... 736
14.3.9. Postprocessing Radiation Quantities ... 736
14.3.9.1. Available Variables for Postprocessing ... 736
14.3.9.2. Reporting Radiative Heat Transfer Through Boundaries ... 737
14.3.9.3. Overall Heat Balances When Using the DTRM ... 737
14.3.9.4. Displaying Rays and Clusters for the DTRM ... 738
14.3.9.4.1. Displaying Clusters ... 738
14.3.9.4.2. Displaying Rays ... 738
14.3.9.4.3. Including the Mesh in the Display ... 739
14.3.9.5. Reporting Radiation in the S2S Model ... 739
14.3.10. Solar Load Model ... 740
14.3.10.1. Introduction ... 741
14.3.10.2. Solar Ray Tracing ... 741
14.3.10.2.1. Shading Algorithm ... 742
14.3.10.2.2. Glazing Materials ... 742
14.3.10.2.3. Inputs ... 743
14.3.10.3. DO Irradiation ... 744
14.3.10.4. Solar Calculator ... 745
14.3.10.4.1. Inputs/Outputs ... 745
14.3.10.4.2. Theory ... 746
14.3.10.4.3. Computation of Load Distribution ... 747
14.3.10.5. Using the Solar Load Model ... 747
14.3.10.5.1. User-Defined Functions (UDFs) for Solar Load ... 748
14.3.10.5.2. Setting Up the Solar Load Model ... 748
14.3.10.5.3. Setting Boundary Conditions for Solar Loading ... 753

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.10.5.4. Solar Ray Tracing ... 754
14.3.10.5.5. DO Irradiation ... 758
14.3.10.5.6. Text Interface-Only Commands ... 759
14.3.10.5.6.1. Automatically Saving Solar Ray Tracing Data ... 759
14.3.10.5.6.2. Automatically Reading Solar Data ... 759
14.3.10.5.6.3. Aligning the Camera Direction With the Position of the Sun ... 760
14.3.10.5.6.4. Specifying the Scattering Fraction ... 760
14.3.10.5.6.5. Applying the Solar Load on Adjacent Fluid Cells ... 760
14.3.10.5.6.6. Specifying Quad Tree Refinement Factor ... 760
14.3.10.5.6.7. Specifying Ground Reflectivity ... 761
14.3.10.5.6.8. Additional Text Interface Commands ... 761
14.3.10.6. Postprocessing Solar Load Quantities ... 761
14.3.10.6.1. Solar Load Animation at Different Sun Positions ... 762
14.3.10.6.2. Reporting and Displaying Solar Load Quantities ... 763
14.4. Modeling Periodic Heat Transfer ... 763
14.4.1. Overview and Limitations ... 764
14.4.1.1. Overview ... 764
14.4.1.2. Constraints for Periodic Heat Transfer Predictions ... 764
14.4.2. Theory ... 764
14.4.2.1. Definition of the Periodic Temperature for Constant- Temperature Wall Conditions ... 765
14.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions ... 765
14.4.3. Using Periodic Heat Transfer ... 765
14.4.4. Solution Strategies for Periodic Heat Transfer ... 767
14.4.5. Monitoring Convergence ... 767
14.4.6. Postprocessing for Periodic Heat Transfer ... 768
15. Modeling Heat Exchangers ... 769
15.1. Overview and Restrictions of the Macro Heat Exchanger Models ... 769
15.1.1. Restrictions ... 770
15.2. Overview and Restrictions of the Dual Cell Model ... 771
15.2.1. Restrictions ... 771
15.3. Using the Ungrouped Macro Heat Exchanger Model ... 771
15.3.1. Selecting the Zone for the Heat Exchanger ... 776
15.3.2. Specifying Heat Exchanger Performance Data ... 776
15.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions ... 777
15.3.4. Defining the Macros ... 777
15.3.4.1. Viewing the Macros ... 778
15.3.5. Specifying the Auxiliary Fluid Properties and Conditions ... 778
15.3.6. Setting the Pressure-Drop Parameters and Effectiveness ... 779
15.3.6.1. Using the Default Core Porosity Model ... 780
15.3.6.2. Defining a New Core Porosity Model ... 780
15.3.6.3. Reading Heat Exchanger Parameters from an External File ... 781
15.3.6.4. Viewing the Parameters for an Existing Core Model ... 781
15.4. Using the Grouped Macro Heat Exchanger Model ... 781
15.4.1. Selecting the Fluid Zones for the Heat Exchanger Group ... 786
15.4.2. Selecting the Upstream Heat Exchanger Group ... 786
15.4.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions ... 786
15.4.4. Specifying the Auxiliary Fluid Properties ... 786
15.4.5. Specifying Supplementary Auxiliary Fluid Streams ... 787
15.4.6. Initializing the Auxiliary Fluid Temperature ... 787
15.5. Using the Dual Cell Heat Exchanger Model ... 787
15.6. Postprocessing for the Heat Exchanger Model ... 794
15.6.1. Total Heat Rejection Rate ... 794

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.6.2. Heat Exchanger Reporting ... 795
15.6.2.1. Computed Heat Rejection ... 795
15.6.2.2. **Inlet/Outlet Temperature** ... 796
15.6.2.3. **Mass Flow Rate** ... 797
15.6.2.4. **Specific Heat** ... 798
15.7. Useful Reporting TUI Commands ... 798
16. Modeling Species Transport and Finite-Rate Chemistry ... 799
16.1. Volumetric Reactions ... 799
16.1.1. Overview of User Inputs for Modeling Species Transport and Reactions ... 799
16.1.1.1. Mixture Materials ... 800
16.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material ... 801
16.1.3. Defining Properties for the Mixture and Its Constituent Species ... 805
16.1.3.1. Defining the Species in the Mixture ... 806
16.1.3.1.1. Overview of the Species Dialog Box ... 807
16.1.3.1.2. Adding Species to the Mixture ... 807
16.1.3.1.3. Removing Species from the Mixture ... 808
16.1.3.1.4. Reordering Species ... 808
16.1.3.1.5. The Naming and Ordering of Species ... 809
16.1.3.2. Defining Reactions ... 809
16.1.3.2.1. Inputs for Reaction Definition ... 809
16.1.3.2.2. Defining Species and Reactions for Fuel Mixtures ... 815
16.1.3.3. Defining Zone-Based Reaction Mechanisms ... 816
16.1.3.3.1. Inputs for Reaction Mechanism Definition ... 816
16.1.3.4. Defining Physical Properties for the Mixture ... 818
16.1.3.5. Defining Physical Properties for the Species in the Mixture ... 819
16.1.4. Setting up Coal Simulations with the Coal Calculator Dialog Box ... 820
16.1.5. Defining Cell Zone and Boundary Conditions for Species ... 822
16.1.5.1. Diffusion at Inlets with the Pressure-Based Solver ... 822
16.1.6. Defining Other Sources of Chemical Species ... 822
16.1.7. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry ... 823
16.1.7.1. Stability and Convergence in Reacting Flows ... 823
16.1.7.2. Two-Step Solution Procedure (Cold Flow Simulation) ... 823
16.1.7.3. Density Under-Relaxation ... 824
16.1.7.4. Ignition in Combustion Simulations ... 824
16.1.7.5. Solution of Stiff Laminar Chemistry Systems ... 824
16.1.7.6. Eddy-Dissipation Concept Model Solution Procedure ... 825
16.1.8. Postprocessing for Species Calculations ... 826
16.1.8.1. Averaged Species Concentrations ... 827
16.1.9. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format ... 827
16.2. Wall Surface Reactions and Chemical Vapor Deposition ... 829
16.2.1. Overview of Surface Species and Wall Surface Reactions ... 829
16.2.2. User Inputs for Wall Surface Reactions ... 829
16.2.3. Including Mass Transfer To Surfaces in Continuity ... 831
16.2.4. Wall Surface Mass Transfer Effects in the Energy Equation ... 831
16.2.5. Modeling the Heat Release Due to Wall Surface Reactions ... 831
16.2.6. Solution Procedures for Wall Surface Reactions ... 831
16.2.7. Postprocessing for Surface Reactions ... 832
16.2.8. Importing a Surface Kinetic Mechanism in CHEMKIN Format ... 832
16.2.8.1. Compatibility and Limitations for Gas Phase Reactions ... 834
16.2.8.2. Compatibility and Limitations for Surface Reactions ... 835
16.3. Particle Surface Reactions ... 835
16.3.1. User Inputs for Particle Surface Reactions ... 835

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.2. Modeling Gaseous Solid Catalyzed Reactions ... 836
16.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion ... 836
16.4. Species Transport Without Reactions ... 837
17. Modeling Non-Premixed Combustion ... 839
17.1. Steps in Using the Non-Premixed Model ... 839
17.1.1. Preliminaries ... 839
17.1.2. Defining the Problem Type ... 840
17.1.3. Overview of the Problem Setup Procedure ... 840
17.2. Setting Up the Equilibrium Chemistry Model ... 843
17.2.1. Choosing Adiabatic or Non-Adiabatic Options ... 844
17.2.2. Specifying the Operating Pressure for the System ... 845
17.2.3. Enabling a Secondary Inlet Stream ... 845
17.2.4. Choosing to Define the Fuel Stream(s) Empirically ... 846
17.2.5. Enabling the Rich Flammability Limit (RFL) Option ... 847
17.3. Setting Up the Steady and Unsteady Laminar Flamelet Models ... 847
17.3.1. Choosing Adiabatic or Non-Adiabatic Options ... 848
17.3.2. Specifying the Operating Pressure for the System ... 848
17.3.3. Specifying a Chemical Mechanism File for Flamelet Generation ... 848
17.3.4. Importing a Flamelet ... 849
17.3.5. Using the Unsteady Laminar Flamelet Model ... 850
17.3.6. Using the Diesel Unsteady Laminar Flamelet Model ... 850
17.4. Defining the Stream Compositions ... 851
17.4.1. Setting Boundary Stream Species ... 853
17.4.1.1. Including Condensed Species ... 854
17.4.2. Modifying the Database ... 854
17.4.3. Composition Inputs for Empirically-Defined Fuel Streams ... 854
17.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model ... 854
17.4.5. Modeling Coal Combustion Using the Non-Premixed Model ... 855
17.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models ... 856
17.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models ... 857
17.4.5.3. Additional Coal Modeling Inputs in ANSYS FLUENT ... 858
17.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion ... 859
17.4.5.5. The Coal Calculator ... 860
17.5. Setting Up Control Parameters ... 861
17.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species ... 862
17.5.2. Defining the Flamelet Controls ... 863
17.5.3. Zeroing Species in the Initial Unsteady Flamelet ... 864
17.6. Calculating the Flamelets ... 864
17.6.1. Steady Flamelet ... 865
17.6.2. Unsteady Flamelet ... 866
17.6.3. Saving the Flamelet Data ... 867
17.6.4. Postprocessing the Flamelet Data ... 868
17.7. Calculating the Look-Up Tables ... 871
17.7.1. Full Tabulation of the Two-Mixture-Fraction Model ... 873
17.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables ... 874
17.7.3. Saving the Look-Up Tables ... 874
17.7.4. Postprocessing the Look-Up Table Data ... 874
17.7.4.1. Files for Flamelet Modeling ... 879
17.7.4.1.1. Standard Flamelet Files ... 879
17.7.4.1.1.1. Sample File ... 879
17.7.4.1.1.2. Missing Species ... 880
17.7.5. Setting Up the Inert Model ... 880

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.7.5.1. Setting Boundary Conditions for Inert Transport ... 882
17.7.5.2. Initializing the Inert Stream ... 882
17.7.5.2.1. Inert Fraction ... 882
17.7.5.2.2. Inert Composition ... 882
17.8. Defining Non-Premixed Boundary Conditions ... 883
17.8.1. Input of Mixture Fraction Boundary Conditions ... 883
17.8.2. Diffusion at Inlets ... 884
17.8.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities ... 885
17.9. Defining Non-Premixed Physical Properties ... 885
17.10. Solution Strategies for Non-Premixed Modeling ... 886
17.10.1. Single-Mixture-Fraction Approach ... 886
17.10.2. Two-Mixture-Fraction Approach ... 886
17.10.3. Starting a Non-Premixed Calculation From a Previous Case File ... 886
17.10.3.1. Retrieving the PDF File During Case File Reads ... 887
17.10.4. Solving the Flow Problem ... 887
17.10.4.1. Under-Relaxation Factors for PDF Equations ... 888
17.10.4.2. Density Under-Relaxation ... 888
17.10.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations ... 888
17.11. Postprocessing the Non-Premixed Model Results ... 889
17.11.1. Postprocessing for Inert Calculations ... 891
18. Modeling Premixed Combustion ... 893
18.1. Overview and Limitations ... 893
18.1.1. Overview ... 893
18.1.2. Limitations ... 894
18.2. Using the Premixed Combustion Model ... 894
18.2.1. Enabling the Premixed Combustion Model ... 895
18.2.2. Choosing an Adiabatic or Non-Adiabatic Model ... 896
18.3. Setting Up the C-Equation and G-Equation Models ... 896
18.3.1. Modifying the Constants for the Zimont Flame Speed Model ... 898
18.3.2. Modifying the Constants for the Peters Flame Speed Model ... 898
18.3.3. Additional Options for the G-Equation Model ... 898
18.3.4. Defining Physical Properties for the Unburnt Mixture ... 898
18.3.5. Setting Boundary Conditions for the Progress Variable ... 899
18.3.6. Initializing the Progress Variable ... 899
18.4. Setting Up the Extended Coherent Flame Model ... 899
18.4.1. Modifying the ECFM Model Variant ... 900
18.4.2. Modifying the Constants for the ECFM Flame Speed Closure ... 900
18.4.3. Setting Boundary Conditions for the ECFM Transport ... 901
18.4.4. Initializing the Flame Area Density ... 901
18.5. Postprocessing for Premixed Combustion Calculations ... 901
18.5.1. Computing Species Concentrations ... 903
19. Modeling Partially Premixed Combustion ... 905
19.1. Overview and Limitations ... 905
19.1.1. Overview ... 905
19.1.2. Limitations ... 905
19.2. Using the Partially Premixed Combustion Model ... 905
19.2.1. Setup and Solution Procedure ... 906
19.2.2. Modifying the Unburnt Mixture Property Polynomials ... 907
20. Modeling a Composition PDF Transport Problem ... 911
20.1. Overview and Limitations ... 911
20.2. Steps for Using the Composition PDF Transport Model ... 911
20.3. Enabling the Lagrangian Composition PDF Transport Model ... 913

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.4. Enabling the Eulerian Composition PDF Transport Model ... 915
20.4.1. Defining Species Boundary Conditions ... 916
20.4.1.1. Equilibrating Inlet Streams ... 917
20.5. Initializing the Solution ... 917
20.6. Monitoring the Solution ... 918
20.6.1. Running Unsteady Composition PDF Transport Simulations ... 919
20.6.2. Running Compressible Lagrangian PDF Transport Simulations ... 920
20.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer ... 920
20.7. Postprocessing for Lagrangian PDF Transport Calculations ... 920
20.7.1. Reporting Options ... 920
20.7.2. Particle Tracking Options ... 921
20.8. Postprocessing for Eulerian PDF Transport Calculations ... 921
20.8.1. Reporting Options ... 921
21. Using Chemistry Acceleration ... 923
21.1. Using ISAT ... 924
21.1.1. ISAT Parameters ... 924
21.1.2. Monitoring ISAT ... 925
21.1.3. Using ISAT Efficiently ... 925
21.1.4. Reading and Writing ISAT Tables ... 926
21.2. Using Chemistry Agglomeration ... 927
21.3. Dimension Reduction ... 927
22. Modeling Engine Ignition ... 929
22.1. Spark Model ... 929
22.1.1. Using the Spark Model ... 929
22.1.2. Using the ECFM Spark Model ... 932
22.2. Autoignition Models ... 933
22.2.1. Using the Autoignition Models ... 933
22.3. Crevice Model ... 936
22.3.1. Using the Crevice Model ... 936
22.3.2. Crevice Model Solution Details ... 938
22.3.3. Postprocessing for the Crevice Model ... 938
22.3.3.1. Using the Crevice Output File ... 940
23. Modeling Pollutant Formation ... 943
23.1. NOx Formation ... 943
23.1.1. Using the NOx Model ... 943
23.1.1.1. Decoupled Analysis: Overview ... 943
23.1.1.2. Enabling the NOx Models ... 944
23.1.1.3. Defining the Fuel Streams ... 946
23.1.1.4. Specifying a User-Defined Function for the NOx Rate ... 948
23.1.1.5. Setting Thermal NOx Parameters ... 948
23.1.1.6. Setting Prompt NOx Parameters ... 949
23.1.1.7. Setting Fuel NOx Parameters ... 949
23.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters ... 950
23.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters ... 950
23.1.1.8. Setting N2O Pathway Parameters ... 952
23.1.1.9. Setting Parameters for NOx Reburn ... 953
23.1.1.10. Setting SNCR Parameters ... 954
23.1.1.11. Setting Turbulence Parameters ... 955
23.1.1.12. Defining Boundary Conditions for the NOx Model ... 958
23.1.2. Solution Strategies ... 958
23.1.3. Postprocessing ... 958
23.2. SOx Formation ... 959

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.1. Using the SOx Model ... 959
23.2.1.1. Enabling the SOx Model ... 960
23.2.1.2. Defining the Fuel Streams ... 961
23.2.1.3. Defining the SOx Fuel Stream Settings ... 963
23.2.1.3.1. Setting SOx Parameters for Gaseous and Liquid Fuel Types ... 964
23.2.1.3.2. Setting SOx Parameters for a Solid Fuel ... 965
23.2.1.4. Setting Turbulence Parameters ... 967
23.2.1.5. Specifying a User-Defined Function for the SOx Rate ... 970
23.2.1.6. Defining Boundary Conditions for the SOx Model ... 970
23.2.2. Solution Strategies ... 971
23.2.3. Postprocessing ... 972
23.3. Soot Formation ... 973
23.3.1. Using the Soot Models ... 973
23.3.1.1. Setting Up the One-Step Model ... 974
23.3.1.2. Setting Up the Two-Step Model ... 975
23.3.1.3. Setting Up the Moss-Brookes Model and the Hall Extension ... 978
23.3.1.3.1. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation ... 981
23.3.1.4. Defining Boundary Conditions for the Soot Model ... 984
23.3.1.5. Reporting Soot Quantities ... 984
23.4. Using the Decoupled Detailed Chemistry Model ... 985
24. Predicting Aerodynamically Generated Noise ... 987
24.1. Overview ... 987
24.1.1. Direct Method ... 987
24.1.2. Integral Method Based on Acoustic Analogy ... 988
24.1.3. Broadband Noise Source Models ... 988
24.2. Using the Ffowcs Williams and Hawkings Acoustics Model ... 989
24.2.1. Enabling the FW-H Acoustics Model ... 990
24.2.1.1. Setting Model Constants ... 991
24.2.1.2. Computing Sound "on the Fly" ... 991
24.2.1.3. Writing Source Data Files ... 992
24.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS FLUENT ASD Format ... 993
24.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format ... 994
24.2.2. Specifying Source Surfaces ... 995
24.2.2.1. Saving Source Data ... 997
24.2.3. Specifying Acoustic Receivers ... 998
24.2.4. Specifying the Time Step ... 999
24.2.5. Postprocessing the FW-H Acoustics Model Data ... 1000
24.2.5.1. Writing Acoustic Signals ... 1000
24.2.5.2. Reading Unsteady Acoustic Source Data ... 1000
24.2.5.2.1. Pruning the Signal Data Automatically ... 1002
24.2.5.3. Reporting the Static Pressure Time Derivative ... 1003
24.2.5.4. Using the FFT Capabilities ... 1003
24.3. Using the Broadband Noise Source Models ... 1003
24.3.1. Enabling the Broadband Noise Source Models ... 1003
24.3.1.1. Setting Model Constants ... 1004
24.3.2. Postprocessing the Broadband Noise Source Model Data ... 1004
25. Modeling Discrete Phase ... 1007
25.1. Introduction ... 1007
25.1.1. Overview ... 1008

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.1.2. Limitations ... 1008
25.1.2.1. Limitation on the Particle Volume Fraction ... 1008
25.1.2.2. Limitation on Modeling Continuous Suspensions of Particles ... 1008
25.1.2.3. Limitations on Using the Discrete Phase Model with Other ANSYS FLUENT Models ... 1008
25.2. Steps for Using the Discrete Phase Models ... 1009
25.2.1. Options for Interaction with the Continuous Phase ... 1010
25.2.2. Steady/Transient Treatment of Particles ... 1010
25.2.3. Tracking Parameters for the Discrete Phase Model ... 1012
25.2.4. Drag Laws ... 1014
25.2.5. Physical Models for the Discrete Phase Model ... 1015
25.2.5.1. Including Radiation Heat Transfer Effects on the Particles ... 1016
25.2.5.2. Including Thermophoretic Force Effects on the Particles ... 1016
25.2.5.3. Including Brownian Motion Effects on the Particles ... 1016
25.2.5.4. Including Saffman Lift Force Effects on the Particles ... 1017
25.2.5.5. Monitoring Erosion/Accretion of Particles at Walls ... 1017
25.2.5.6. Including the Effect of Particles on Turbulent Quantities ... 1017
25.2.6. Options for Spray Modeling ... 1017
25.2.6.1. Modeling Droplet Collisions ... 1017
25.2.6.2. Modeling Spray Breakup ... 1017
25.2.7. User-Defined Functions ... 1018
25.2.8. Numerics of the Discrete Phase Model ... 1019
25.2.8.1. Numerics for Tracking of the Particles ... 1020
25.2.8.2. Including Coupled Heat-Mass Solution Effects on the Particles ... 1021
25.2.8.3. Tracking in a Reference Frame ... 1021
25.2.8.4. Staggering of Particles in Space and Time ... 1022
25.3. Setting Initial Conditions for the Discrete Phase ... 1022
25.3.1. Injection Types ... 1024
25.3.2. Particle Types ... 1025
25.3.3. Point Properties for Single Injections ... 1026
25.3.4. Point Properties for Group Injections ... 1027
25.3.5. Point Properties for Cone Injections ... 1027
25.3.6. Point Properties for Surface Injections ... 1029
25.3.6.1. Using the Rosin-Rammler Diameter Distribution Method ... 1030
25.3.7. Point Properties for Plain-Orifice Atomizer Injections ... 1030
25.3.8. Point Properties for Pressure-Swirl Atomizer Injections ... 1031
25.3.9. Point Properties for Air-Blast/Air-Assist Atomizer Injections ... 1032
25.3.10. Point Properties for Flat-Fan Atomizer Injections ... 1033
25.3.11. Point Properties for Effervescent Atomizer Injections ... 1034
25.3.12. Point Properties for File Injections ... 1034
25.3.13. Using the Rosin-Rammler Diameter Distribution Method ... 1035
25.3.13.1. The Stochastic Rosin-Rammler Diameter Distribution Method ... 1038
25.3.14. Creating and Modifying Injections ... 1038
25.3.14.1. Creating Injections ... 1039
25.3.14.2. Modifying Injections ... 1039
25.3.14.3. Copying Injections ... 1039
25.3.14.4. Deleting Injections ... 1039
25.3.14.5. Listing Injections ... 1040
25.3.14.6. Reading and Writing Injections ... 1040
25.3.14.7. Shortcuts for Selecting Injections ... 1040
25.3.15. Defining Injection Properties ... 1040
25.3.16. Specifying Turbulent Dispersion of Particles ... 1044
25.3.16.1. Stochastic Tracking ... 1044

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.16.2. Cloud Tracking 1045
25.3.17. Custom Particle Laws 1046
25.3.18. Defining Properties Common to More than One Injection 1047
25.3.18.1. Modifying Properties 1047
25.3.18.2. Modifying Properties Common to a Subset of Selected Injections 1049
25.3.19. Point Properties for Transient Injections 1049
25.4. Setting Boundary Conditions for the Discrete Phase 1050
25.4.1. Discrete Phase Boundary Condition Types 1050
25.4.1.1. Default Discrete Phase Boundary Conditions 1053
25.4.2. Setting Particle Erosion and Accretion Parameters 1054
25.5. Setting Material Properties for the Discrete Phase 1054
25.5.1. Summary of Property Inputs 1054
25.5.2. Setting Discrete-Phase Physical Properties 1057
25.5.2.1. The Concept of Discrete-Phase Materials 1057
25.5.2.1.1. Defining Additional Discrete-Phase Materials 1058
25.5.2.2. Description of the Properties 1059
25.6. Solution Strategies for the Discrete Phase 1063
25.6.1. Performing Trajectory Calculations 1064
25.6.1.1. Uncoupled Calculations 1064
25.6.1.2. Coupled Calculations 1064
25.6.1.2.1. Procedures for a Coupled Two-Phase Flow 1065
25.6.1.2.2. Stochastic Tracking in Coupled Calculations 1066
25.6.1.2.3. Under-Relaxation of the Interphase Exchange Terms 1066
25.6.2. Resetting the Interphase Exchange Terms 1067
25.7. Postprocessing for the Discrete Phase 1067
25.7.1. Displaying of Trajectories 1068
25.7.1.1. Specifying Individual Particles for Display 1070
25.7.1.2. Importing Particle Data 1070
25.7.1.3. Options for Particle Trajectory Plots 1071
25.7.1.4. Graphical Display for Axisymmetric Geometries 1071
25.7.2. Reporting of Trajectory Fates 1072
25.7.2.1. Trajectory Fates 1072
25.7.2.2. Summary Reports 1072
25.7.2.2.1. Elapsed Time 1074
25.7.2.2.2. Mass Transfer Summary 1074
25.7.2.2.3. Energy Transfer Summary 1074
25.7.2.2.4. Heat Rate and Energy Reporting 1075
25.7.2.2.4.1. Change of Heat and Change of Energy Reporting 1076
25.7.2.2.5. Combusting Particles 1077
25.7.2.2.6. Combusting Particles with the Multiple Surface Reaction Model 1077
25.7.2.2.7. Multicomponent Particles 1078
25.7.3. Step-by-Step Reporting of Trajectories 1078
25.7.4. Reporting of Current Positions for Unsteady Tracking 1080
25.7.5. Reporting of Interphase Exchange Terms and Discrete Phase Concentration 1081
25.7.6. Sampling of Trajectories 1082
25.7.7. Histogram Reporting of Samples 1084
25.7.8. Summary Reporting of Current Particles 1085
25.7.9. Postprocessing of Erosion/Accretion Rates 1087
25.8. Parallel Processing for the Discrete Phase Model 1087
26. Modeling Multiphase Flows 1091
26.1. Introduction 1091
26.2. Steps for Using a Multiphase Model 1091

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.1. Enabling the Multiphase Model ... 1093
26.2.2. Choosing a Volume Fraction Formulation ... 1095
26.2.2.1. Explicit Schemes ... 1095
26.2.2.2. Implicit Schemes ... 1097
26.2.2.2.1. Examples ... 1097
26.2.2.3. Volume Fraction Limits ... 1097
26.2.3. Solving a Homogeneous Multiphase Flow ... 1098
26.2.4. Defining the Phases ... 1098
26.2.5. Including Body Forces ... 1099
26.2.6. Modeling Multiphase Species Transport ... 1100
26.2.7. Specifying Heterogeneous Reactions ... 1101
26.2.8. Including Mass Transfer Effects ... 1104
26.2.9. Defining Multiphase Cell Zone and Boundary Conditions ... 1107
26.2.9.1. Boundary Conditions for the Mixture and the Individual Phases ... 1108
26.2.9.1.1. VOF Model ... 1108
26.2.9.1.2. Mixture Model ... 1109
26.2.9.1.3. Eulerian Model ... 1111
26.2.9.2. Steps for Setting Boundary Conditions ... 1115
26.2.9.3. Steps for Copying Cell Zone and Boundary Conditions ... 1119
26.3. Setting Up the VOF Model ... 1120
26.3.1. Including Coupled Level Set with the VOF Model ... 1120
26.3.2. Modeling Open Channel Flows ... 1121
26.3.2.1. Defining Inlet Groups ... 1121
26.3.2.2. Defining Outlet Groups ... 1122
26.3.2.3. Setting the Inlet Group ... 1122
26.3.2.4. Setting the Outlet Group ... 1122
26.3.2.5. Determining the Free Surface Level ... 1123
26.3.2.6. Determining the Bottom Level ... 1123
26.3.2.7. Specifying the Total Height ... 1124
26.3.2.8. Determining the Velocity Magnitude ... 1124
26.3.2.9. Determining the Secondary Phase for the Inlet ... 1124
26.3.2.10. Choosing the Pressure Specification Method ... 1125
26.3.2.11. Limitations ... 1125
26.3.2.12. Choosing the Density Interpolation Method ... 1126
26.3.2.13. Recommendations for Setting Up an Open Channel Flow Problem ... 1126
26.3.3. Modeling Open Channel Wave Boundary Conditions ... 1127
26.3.4. Recommendations for Open Channel Initialization ... 1130
26.3.5. Numerical Beach Treatment for Open Channels ... 1133
26.3.5.1. Solution Strategies ... 1135
26.3.6. Defining the Phases for the VOF Model ... 1136
26.3.6.1. Defining the Primary Phase ... 1136
26.3.6.2. Defining a Secondary Phase ... 1137
26.3.6.3. Including Surface Tension and Adhesion Effects ... 1138
26.3.6.4. Discretizing Using the Phase Localized Compressive Scheme ... 1141
26.3.7. Setting Time-Dependent Parameters for the VOF Model ... 1143
26.3.8. Modeling Compressible Flows ... 1144
26.3.9. Modeling Solidification/Melting ... 1145
26.4. Setting Up the Mixture Model ... 1145
26.4.1. Defining the Phases for the Mixture Model ... 1145
26.4.1.1. Defining the Primary Phase ... 1145
26.4.1.2. Defining a Nongranular Secondary Phase ... 1146
26.4.1.3. Defining a Granular Secondary Phase ... 1147

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.4.1.4. Defining the Interfacial Area Concentration 1149
26.4.1.5. Defining Drag Between Phases 1151
26.4.1.6. Defining the Slip Velocity 1152
26.4.2. Including Cavitation Effects 1153
26.4.3. Modeling Compressible Flows 1153
26.5. Setting Up the Eulerian Model 1154
26.5.1. Additional Guidelines for Eulerian Multiphase Simulations 1154
26.5.2. Defining the Phases for the Eulerian Model 1154
26.5.2.1. Defining the Primary Phase 1154
26.5.2.2. Defining a Nongranular Secondary Phase 1154
26.5.2.3. Defining a Granular Secondary Phase 1155
26.5.2.4. Defining the Interfacial Area Concentration 1158
26.5.2.5. Defining the Interaction Between Phases 1160
26.5.2.5.1. Specifying the Drag Function 1160
26.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only) 1161
26.5.2.6. Including the Lift Force 1161
26.5.2.7. Including Surface Tension and Wall Adhesion Effects 1162
26.5.2.7.1. Including the Virtual Mass Force 1163
26.5.3. Setting Time-Dependent Parameters for the Explicit Volume Fraction Scheme 1163
26.5.4. Modeling Turbulence 1163
26.5.4.1. Including Source Terms 1165
26.5.4.2. Customizing the k-ε Multiphase Turbulent Viscosity 1165
26.5.5. Including Heat Transfer Effects 1165
26.5.6. Modeling Compressible Flows 1166
26.5.7. Including the Dense Discrete Phase Model 1167
26.5.7.1. Defining a Granular Discrete Phase 1170
26.5.8. Including the Boiling Model 1171
26.5.9. Including the Immiscible Fluid Model 1176
26.6. Setting Up the Wet Steam Model 1178
26.6.1. Using User-Defined Thermodynamic Wet Steam Properties 1178
26.6.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF) 1179
26.6.3. Compiling Your UDWSPF and Building a Shared Library File 1181
26.6.4. Loading the UDWSPF Shared Library File 1182
26.6.5. UDWSPF Example 1183
26.7. Solution Strategies for Multiphase Modeling 1187
26.7.1. Coupled Solution for Multiphase Flows 1187
26.7.2. Setting Initial Volume Fractions 1189
26.7.3. VOF Model 1189
26.7.3.1. Setting the Reference Pressure Location 1189
26.7.3.2. Pressure Interpolation Scheme 1189
26.7.3.3. Discretization Scheme Selection for the Implicit and Explicit Formulations 1190
26.7.3.4. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations 1191
26.7.3.5. Under-Relaxation for the Steady-State Formulation 1191
26.7.4. Mixture Model 1192
26.7.4.1. Setting the Under-Relaxation Factor for the Slip Velocity 1192
26.7.4.2. Calculating an Initial Solution 1192
26.7.5. Eulerian Model 1192
26.7.5.1. Calculating an Initial Solution 1192
26.7.5.2. Temporarily Ignoring Lift and Virtual Mass Forces 1193
26.7.5.3. Selecting the Solution Method 1193
26.7.5.4. Discretization Scheme Selection for the Implicit and Explicit Formulations 1193

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.7.5.5. Using W-Cycle Multigrid ... 1194
26.7.5.6. Including the Anisotropic Drag Law ... 1194
26.7.6. Wet Steam Model ... 1195
26.7.6.1. Boundary Conditions, Initialization, and Patching ... 1195
26.7.6.2. Solution Limits for the Wet Steam Model ... 1195
26.7.6.3. Solution Strategies for the Wet Steam Model ... 1195
26.8. Postprocessing for Multiphase Modeling ... 1196
26.8.1. Model-Specific Variables ... 1196
26.8.1.1. VOF Model ... 1196
26.8.1.2. Mixture Model ... 1196
26.8.1.3. Eulerian Model ... 1197
26.8.1.4. Multiphase Species Transport ... 1198
26.8.1.5. Wet Steam Model ... 1198
26.8.1.6. Dense Discrete Phase Model ... 1199
26.8.2. Displaying Velocity Vectors ... 1199
26.8.3. Reporting Fluxes ... 1200
26.8.4. Reporting Forces on Walls ... 1200
26.8.5. Reporting Flow Rates ... 1200
27. Modeling Solidification and Melting ... 1201
27.1. Setup Procedure ... 1201
27.2. Procedures for Modeling Continuous Casting ... 1204
27.3. Solution Procedure ... 1205
27.4. Postprocessing ... 1205
28. Using the Solver ... 1207
28.1. Overview of Using the Solver ... 1207
28.1.1. Choosing the Solver ... 1209
28.2. Choosing the Spatial Discretization Scheme ... 1210
28.2.1. First-Order Accuracy vs. Second-Order Accuracy ... 1211
28.2.1.1. First-to-Higher Order Blending ... 1211
28.2.2. Other Discretization Schemes ... 1212
28.2.3. Choosing the Pressure Interpolation Scheme ... 1212
28.2.4. Choosing the Density Interpolation Scheme ... 1213
28.2.5. User Inputs ... 1213
28.3. Pressure-Based Solver Settings ... 1215
28.3.1. Choosing the Pressure-Velocity Coupling Method ... 1215
28.3.1.1. SIMPLE vs. SIMPLEC ... 1216
28.3.1.2. PISO ... 1216
28.3.1.3. Fractional Step Method ... 1216
28.3.1.4. Coupled ... 1217
28.3.1.4.1. User Inputs ... 1217
28.3.2. Setting Under-Relaxation Factors ... 1217
28.3.2.1. User Inputs ... 1218
28.3.3. Setting Solution Controls for the Non-Iterative Solver ... 1219
28.3.3.1. User Inputs ... 1220
28.4. Density-Based Solver Settings ... 1222
28.4.1. Changing the Courant Number ... 1222
28.4.1.1. Courant Numbers for the Density-Based Explicit Formulation ... 1223
28.4.1.2. Courant Numbers for the Density-Based Implicit Formulation ... 1223
28.4.1.3. User Inputs ... 1223
28.4.2. Convective Flux Types ... 1224
28.4.2.1. User Inputs ... 1225
28.4.3. Specifying the Explicit Relaxation ... 1225

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.4. Turning On FAS Multigrid ... 1225
28.4.4.1. Setting Coarse Grid Levels ... 1225
28.4.4.2. Using Residual Smoothing to Increase the Courant Number ... 1226
28.5. Setting Algebraic Multigrid Parameters ... 1226
28.5.1. Specifying the Multigrid Cycle Type ... 1227
28.5.2. Setting the Termination and Residual Reduction Parameters ... 1227
28.5.3. Setting the AMG Method and the Stabilization Parameters ... 1228
28.5.4. Additional Algebraic Multigrid Parameters ... 1228
28.5.4.1. Fixed Cycle Parameters ... 1229
28.5.4.2. Coarsening Parameters ... 1229
28.5.4.3. Smoother Types ... 1230
28.5.4.4. Flexible Cycle Parameters ... 1230
28.5.4.5. Setting the Verbosity ... 1231
28.5.4.6. Returning to the Default Multigrid Parameters ... 1232
28.5.5. Setting FAS Multigrid Parameters ... 1232
28.5.5.1. Combating Convergence Trouble ... 1232
28.5.5.2. "Industrial-Strength" FAS Multigrid ... 1232
28.6. Setting Solution Limits ... 1235
28.6.1. Limiting the Values of Solution Variables ... 1236
28.6.2. Adjusting the Positivity Rate Limit ... 1236
28.6.3. Resetting Solution Limits ... 1237
28.7. Setting Multi-Stage Time-Stepping Parameters ... 1237
28.7.1. Changing the Multi-Stage Scheme ... 1237
28.7.1.1. Changing the Coefficients and Number of Stages ... 1238
28.7.1.2. Controlling Updates to Dissipation and Viscous Stresses ... 1238
28.7.1.3. Resetting the Multi-Stage Parameters ... 1239
28.8. Selecting Gradient Limiters ... 1239
28.9. Initializing the Solution ... 1239
28.9.1. Initializing the Entire Flow Field Using Standard Initialization ... 1240
28.9.1.1. Saving and Resetting Initial Values ... 1242
28.9.2. Patching Values in Selected Cells ... 1242
28.9.2.1. Using Registers ... 1243
28.9.2.2. Using Field Functions ... 1243
28.9.2.3. Using Patching Later in the Solution Process ... 1243
28.10. Full Multigrid (FMG) Initialization ... 1244
28.10.1. Steps in Using FMG Initialization ... 1244
28.10.2. Convergence Strategies for FMG Initialization ... 1245
28.11. Hybrid Initialization ... 1245
28.11.1. Steps in Using Hybrid Initialization ... 1245
28.11.2. Solution Strategies for Hybrid Initialization ... 1247
28.12. Performing Steady-State Calculations ... 1247
28.12.1. Updating UDF Profiles ... 1248
28.12.2. Interrupting Iterations ... 1249
28.12.3. Resetting Data ... 1249
28.13. Performing Pseudo Transient Calculations ... 1249
28.13.1. Setting Pseudo Transient Explicit Relaxation Factors ... 1250
28.13.1.1. User Inputs ... 1250
28.13.2. Setting Solution Controls for the Pseudo Transient Method ... 1251
28.13.3. Solving Pseudo-Transient Flow ... 1252
28.14. Performing Time-Dependent Calculations ... 1255
28.14.1. User Inputs for Time-Dependent Problems ... 1256
28.14.1.1. Additional Inputs ... 1264

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.14.2. Adaptive Time Stepping ... 1264
28.14.2.1. The Adaptive Time Stepping Algorithm ... 1264
28.14.2.2. Specifying Parameters for Adaptive Time Stepping ... 1265
28.14.2.3. Specifying a User-Defined Time Stepping Method ... 1266
28.14.3. Variable Time Stepping ... 1266
28.14.3.1. The Variable Time Stepping Algorithm ... 1266
28.14.3.2. Specifying Parameters for Variable Time Stepping ... 1267
28.14.4. Postprocessing for Time-Dependent Problems ... 1267
28.15. Monitoring Solution Convergence ... 1268
28.15.1. Monitoring Residuals ... 1269
28.15.1.1. Definition of Residuals for the Pressure-Based Solver ... 1269
28.15.1.2. Definition of Residuals for the Density-Based Solver ... 1270
28.15.1.3. Overview of Using the Residual Monitors Dialog Box ... 1272
28.15.1.4. Printing and Plotting Residuals ... 1272
28.15.1.5. Storing Residual History Points ... 1273
28.15.1.6. Controlling Normalization and Scaling ... 1273
28.15.1.7. Choosing a Convergence Criterion ... 1274
28.15.1.8. Modifying Convergence Criteria ... 1276
28.15.1.9. Disabling Monitoring ... 1276
28.15.1.10. Plot Parameters ... 1276
28.15.1.11. Postprocessing Residual Values ... 1277
28.15.2. Monitoring Statistics ... 1277
28.15.2.1. Plot Parameters ... 1278
28.15.3. Monitoring Force and Moment Coefficients ... 1278
28.15.3.1. Setting Up Force and Moment Coefficient Monitors ... 1279
28.15.3.1.1. Specifying the Reporting Methods ... 1282
28.15.3.1.1.1. Plot Parameters ... 1282
28.15.3.1.2. Monitoring Individual Walls ... 1282
28.15.3.1.3. Faster Force and Moment Monitoring ... 1283
28.15.3.1.4. Discarding the Monitor Data ... 1283
28.15.4. Monitoring Surface Integrals ... 1283
28.15.4.1. Overview of Defining Surface Monitors ... 1283
28.15.4.2. Printing, Plotting, and Saving Surface Integration Histories ... 1285
28.15.4.2.1. Plot Parameters ... 1286
28.15.5. Monitoring Volume Integrals ... 1286
28.15.5.1. Overview of Defining Volume Monitors ... 1286
28.15.5.2. Printing, Plotting, and Saving Volume Integration Histories ... 1288
28.15.5.2.1. Plot Parameters ... 1288
28.16. Executing Commands During the Calculation ... 1288
28.16.1. Defining Macros ... 1290
28.16.2. Saving Files During the Calculation ... 1291
28.17. Automatic Initialization of the Solution and Case Modification ... 1292
28.17.1. Altering the Solution Initialization and Case Modification after Calculating ... 1295
28.18. Animating the Solution ... 1296
28.18.1. Defining an Animation Sequence ... 1296
28.18.1.1. Guidelines for Defining an Animation Sequence ... 1299
28.18.2. Playing an Animation Sequence ... 1300
28.18.2.1. Modifying the View ... 1300
28.18.2.2. Modifying the Playback Speed ... 1301
28.18.2.3. Playing Back an Excerpt ... 1301
28.18.2.4. “Fast-Forwarding” the Animation ... 1301
28.18.2.5. Continuous Animation ... 1301

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.18.2.6. Stopping the Animation ... 1301
28.18.2.7. Advancing the Animation Frame by Frame ... 1301
28.18.2.8. Deleting an Animation Sequence ... 1302
28.18.3. Saving an Animation Sequence ... 1302
28.18.3.1. Solution Animation File ... 1302
28.18.3.2. Picture File ... 1303
28.18.3.3. MPEG File ... 1303
28.18.4. Reading an Animation Sequence ... 1303
28.19. Checking Your Case Setup ... 1304
28.19.1. Automatic Implementation ... 1305
28.19.2. Manual Implementation ... 1305
28.19.2.1. Checking the Mesh ... 1306
28.19.2.2. Checking Model Selections ... 1308
28.19.2.3. Checking Boundary and Cell Zone Conditions ... 1310
28.19.2.4. Checking Material Properties ... 1313
28.19.2.5. Checking the Solver Settings ... 1314
28.20. Convergence and Stability ... 1317
28.20.1. Judging Convergence ... 1317
28.20.2. Step-by-Step Solution Processes ... 1318
28.20.2.1. Selecting a Subset of the Solution Equations ... 1318
28.20.2.2. Turning Reactions On and Off ... 1319
28.20.3. Modifying Algebraic Multigrid Parameters ... 1319
28.20.4. Modifying the Multi-Stage Parameters ... 1319
28.21. Solution Steering ... 1320
28.21.1. Overview of Solution Steering ... 1320
28.21.2. Solution Steering Strategy ... 1320
28.21.2.1. Initialization ... 1320
28.21.3. Using Solution Steering ... 1321
29. Adapting the Mesh ... 1325
29.1. Using Adaption ... 1325
29.1.1. Adaption Example ... 1326
29.1.2. Adaption Guidelines ... 1327
29.2. Boundary Adaption ... 1328
29.2.1. Performing Boundary Adaption ... 1329
29.2.1.1. Boundary Adaption Based on Number of Cells ... 1329
29.2.1.2. Boundary Adaption Based on Normal Distance ... 1330
29.2.1.3. Boundary Adaption Based on Target Boundary Volume ... 1330
29.3. Gradient Adaption ... 1331
29.3.1. Performing Gradient Adaption ... 1331
29.4. Dynamic Gradient Adaption ... 1333
29.4.1. Dynamic Gradient Adaption Approach ... 1334
29.4.1.1. Examples of Dynamic Gradient Adaption ... 1335
29.5. Isovalue Adaption ... 1335
29.5.1. Performing Isovalue Adaption ... 1335
29.6. Region Adaption ... 1336
29.6.1. Performing Region Adaption ... 1336
29.7. Volume Adaption ... 1337
29.7.1. Performing Volume Adaption ... 1337
29.8. Yplus/Ystar Adaption ... 1338
29.8.1. Performing Yplus or Ystar Adaption ... 1338
29.9. Anisotropic Adaption ... 1339
29.9.1. Limitations of Anisotropic Adaption ... 1340

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.9.2. Performing Anisotropic Adaption ... 1340
29.10. Geometry-Based Adaption ... 1341
29.10.1. Performing Geometry-Based Adaption ... 1341
29.11. Registers ... 1342
29.11.1. Manipulating Adaption Registers ... 1343
29.11.1.1. Changing Register Types ... 1343
29.11.1.2. Combining Registers ... 1344
29.11.1.3. Deleting Registers ... 1345
29.11.2. Modifying Adaption Marks ... 1345
29.11.3. Displaying Registers ... 1346
29.11.3.1. Adaption Display Options ... 1346
29.11.4. Adapting to Registers ... 1347
29.12. Mesh Adaption Controls ... 1347
29.12.1. Limiting Adaption by Zone ... 1348
29.12.2. Limiting Adaption by Cell Volume or Volume Weight ... 1349
29.12.3. Limiting the Total Number of Cells ... 1349
29.12.4. Controlling the Levels of Refinement During Hanging Node Adaption ... 1349
29.13. Improving the Mesh by Smoothing and Swapping ... 1349
29.13.1. Smoothing ... 1350
29.13.1.1. Quality-Based Smoothing ... 1351
29.13.1.2. Laplacian Smoothing ... 1351
29.13.1.3. Skewness-Based Smoothing ... 1353
29.13.2. Face Swapping ... 1353
29.13.2.1. Triangular Meshes ... 1353
29.13.2.2. Tetrahedral Meshes ... 1354
29.13.3. Combining Skewness-Based Smoothing and Face Swapping ... 1354
30. Creating Surfaces for Displaying and Reporting Data ... 1355
30.1. Using Surfaces ... 1355
30.2. Zone Surfaces ... 1356
30.3. Partition Surfaces ... 1357
30.4. Point Surfaces ... 1359
30.4.1. Using the Point Tool ... 1360
30.4.1.1. Initializing the Point Tool ... 1360
30.4.1.2. Translating the Point Tool ... 1361
30.4.1.3. Resetting the Point Tool ... 1361
30.5. Line and Rake Surfaces ... 1361
30.5.1. Using the Line Tool ... 1363
30.5.1.1. Initializing the Line Tool ... 1363
30.5.1.2. Translating the Line Tool ... 1364
30.5.1.3. Rotating the Line Tool ... 1364
30.5.1.4. Resizing the Line Tool ... 1364
30.5.1.5. Resetting the Line Tool ... 1364
30.6. Plane Surfaces ... 1364
30.6.1. Using the Plane Tool ... 1367
30.6.1.1. Initializing the Plane Tool ... 1367
30.6.1.2. Translating the Plane Tool ... 1368
30.6.1.3. Rotating the Plane Tool ... 1368
30.6.1.4. Resizing the Plane Tool ... 1369
30.6.1.5. Resetting the Plane Tool ... 1369
30.7. Quadric Surfaces ... 1369
30.8. Isosurfaces ... 1371
30.9. Clipping Surfaces ... 1373

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.10. Transforming Surfaces 1375
30.11. Grouping, Renaming, and Deleting Surfaces 1376
30.11.1. Grouping Surfaces 1377
30.11.2. Renaming Surfaces 1377
30.11.3. Deleting Surfaces 1378
30.11.4. Surface Statistics 1378
31. Displaying Graphics 1379
31.1. Basic Graphics Generation 1379
31.1.1. Displaying the Mesh 1379
31.1.1.1. Generating Mesh or Outline Plots 1381
31.1.1.2. Mesh and Outline Display Options 1383
31.1.1.2.1. Modifying the Mesh Colors 1383
31.1.1.2.2. Adding Features to an Outline Display 1384
31.1.1.2.3. Drawing Partition Boundaries 1385
31.1.1.2.4. Shrinking Faces and Cells in the Display 1385
31.1.2. Displaying Contours and Profiles 1386
31.1.2.1. Generating Contour and Profile Plots 1387
31.1.2.2. Contour and Profile Plot Options 1389
31.1.2.2.1. Drawing Filled Contours or Profiles 1390
31.1.2.2.2. Specifying the Range of Magnitudes Displayed 1390
31.1.2.2.3. Including the Mesh in the Contour Plot 1392
31.1.2.2.4. Choosing Node or Cell Values 1392
31.1.2.2.5. Storing Contour Plot Settings 1393
31.1.3. Displaying Vectors 1393
31.1.3.1. Generating Vector Plots 1394
31.1.3.2. Displaying Relative Velocity Vectors 1395
31.1.3.3. Vector Plot Options 1395
31.1.3.3.1. Scaling the Vectors 1395
31.1.3.3.2. Skipping Vectors 1396
31.1.3.3.3. Drawing Vectors in the Plane of the Surface 1396
31.1.3.3.4. Displaying Fixed-Length Vectors 1397
31.1.3.3.5. Displaying Vector Components 1397
31.1.3.3.6. Specifying the Range of Magnitudes Displayed 1397
31.1.3.3.7. Changing the Scalar Field Used for Coloring the Vectors 1397
31.1.3.3.8. Displaying Vectors Using a Single Color 1398
31.1.3.3.9. Including the Mesh in the Vector Plot 1398
31.1.3.3.10. Changing the Arrow Characteristics 1398
31.1.3.4. Creating and Managing Custom Vectors 1398
31.1.3.4.1. Creating Custom Vectors 1398
31.1.3.4.2. Manipulating, Saving, and Loading Custom Vectors 1399
31.1.4. Displaying Pathlines 1400
31.1.4.1. Steps for Generating Pathlines 1401
31.1.4.2. Options for Pathline Plots 1402
31.1.4.2.1. Including the Mesh in the Pathline Display 1403
31.1.4.2.2. Controlling the Pathline Style 1403
31.1.4.2.3. Controlling Pathline Colors 1404
31.1.4.2.4. "Thinning" Pathlines 1404
31.1.4.2.5. Coarsening Pathlines 1404
31.1.4.2.6. Reversing the Pathlines 1404
31.1.4.2.7. Plotting Oil-Flow Pathlines 1405
31.1.4.2.8. Controlling the Pulse Mode 1405
31.1.4.2.9. Controlling the Accuracy 1405

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.4.2.10. Plotting Relative Pathlines ... 1405
31.1.4.2.11. Generating an XY Plot Along Pathline Trajectories ... 1405
31.1.4.2.12. Saving Pathline Data ... 1405
31.1.4.2.12.1. Standard Type ... 1406
31.1.4.2.12.2. Geometry Type ... 1407
31.1.4.2.12.3. EnSight Type ... 1407
31.1.4.2.13. Choosing Node or Cell Values ... 1409
31.1.5. Displaying Results on a Sweep Surface ... 1409
31.1.5.1. Steps for Generating a Plot Using a Sweep Surface ... 1409
31.1.5.2. Animating a Sweep Surface Display ... 1410
31.1.6. Hiding the Graphics Window Display ... 1410
31.2. Customizing the Graphics Display ... 1411
31.2.1. Overlay of Graphics ... 1411
31.2.2. Opening Multiple Graphics Windows ... 1412
31.2.2.1. Setting the Active Window ... 1413
31.2.3. Changing the Legend Display ... 1413
31.2.3.1. Enabling/Disabling the Legend, Logo, and Color Scale ... 1414
31.2.3.2. Editing the Legend ... 1414
31.2.3.3. Adding a Title to the Caption ... 1414
31.2.3.4. Enabling/Disabling the Axes ... 1414
31.2.3.5. Displaying/Hiding the Logo ... 1414
31.2.3.6. Colormap Alignment ... 1415
31.2.4. Adding Text to the Graphics Window ... 1415
31.2.4.1. Adding Text Using the Annotate Dialog Box ... 1415
31.2.4.2. Adding Text Using the **Mouse-Annotate** Function ... 1416
31.2.4.3. Editing Existing Annotation Text ... 1417
31.2.4.4. Clearing Annotation Text ... 1417
31.2.5. Changing the Colormap ... 1417
31.2.5.1. Predefined Colormaps ... 1418
31.2.5.2. Selecting a Colormap ... 1419
31.2.5.2.1. Specifying the Colormap Size and Scale ... 1419
31.2.5.2.2. Changing the Number Format ... 1419
31.2.5.3. Displaying Colormap Label ... 1419
31.2.5.4. Creating a Customized Colormap ... 1421
31.2.6. Adding Lights ... 1423
31.2.6.1. Turning on Lighting Effects with the Display Options Dialog Box ... 1423
31.2.6.2. Turning on Lighting Effects with the **Lights** Dialog Box ... 1423
31.2.6.3. Defining Light Sources ... 1424
31.2.6.3.1. Removing a Light ... 1425
31.2.6.3.2. Resetting the Light Definitions ... 1425
31.2.7. Modifying the Rendering Options ... 1425
31.2.7.1. Graphics Device Information ... 1426
31.3. Controlling the Mouse Button Functions ... 1427
31.3.1. Button Functions ... 1427
31.3.2. Modifying the Mouse Button Functions ... 1428
31.4. Viewing the Application Window ... 1428
31.4.1. Embedding the Graphics Windows ... 1432
31.5. Modifying the View ... 1433
31.5.1. Selecting a View ... 1434
31.5.2. Manipulating the Display ... 1434
31.5.2.1. Scaling and Centering ... 1435
31.5.2.2. Rotating the Display ... 1435

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.2.2.1. Spinning the Display with the Mouse 1436
31.5.2.3. Translating the Display 1437
31.5.2.4. Zooming the Display 1437
31.5.3. Controlling Perspective and Camera Parameters 1438
31.5.3.1. Perspective and Orthographic Views 1438
31.5.3.2. Modifying Camera Parameters 1438
31.5.4. Saving and Restoring Views 1439
31.5.4.1. Restoring the Default View 1439
31.5.4.2. Returning to Previous Views 1439
31.5.4.3. Saving Views 1440
31.5.4.4. Reading View Files 1440
31.5.4.5. Deleting Views 1440
31.5.5. Mirroring and Periodic Repeats 1441
31.5.5.1. Periodic Repeats for Graphics 1443
31.5.5.2. Mirroring for Graphics 1444
31.6. Composing a Scene 1444
31.6.1. Selecting the Object(s) to be Manipulated 1445
31.6.2. Changing an Object's Display Properties 1446
31.6.2.1. Controlling Visibility 1446
31.6.2.2. Controlling Object Color and Transparency 1447
31.6.3. Transforming Geometric Objects in a Scene 1448
31.6.3.1. Translating Objects 1449
31.6.3.2. Rotating Objects 1449
31.6.3.3. Scaling Objects 1449
31.6.3.4. Displaying the Meridional View 1449
31.6.4. Modifying Iso-Values 1450
31.6.4.1. Steps for Modifying Iso-Values 1450
31.6.4.2. An Example of Iso-Value Modification for an Animation 1450
31.6.5. Modifying Pathline Attributes 1451
31.6.5.1. An Example of Pathline Modification for an Animation 1451
31.6.6. Deleting an Object from the Scene 1452
31.6.7. Adding a Bounding Frame 1452
31.7. Animating Graphics 1454
31.7.1. Creating an Animation 1455
31.7.1.1. Deleting Key Frames 1455
31.7.2. Playing an Animation 1455
31.7.2.1. Playing Back an Excerpt 1456
31.7.2.2. "Fast-Forwarding" the Animation 1456
31.7.2.3. Continuous Animation 1456
31.7.2.4. Stopping the Animation 1456
31.7.2.5. Advancing the Animation Frame by Frame 1456
31.7.3. Saving an Animation 1456
31.7.3.1. Animation File 1457
31.7.3.2. Picture File 1457
31.7.3.3. MPEG File 1457
31.7.4. Reading an Animation File 1458
31.7.5. Notes on Animation 1458
31.8. Creating Videos 1458
31.8.1. Recording Animations To Video 1458
31.8.1.1. Computer Image vs. Video Image 1459
31.8.1.2. Real-Time vs. Frame-By-Frame 1459
31.8.2. Equipment Required 1459

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.3. Recording an Animation with ANSYS FLUENT ... 1460
31.8.3.1. Create an Animation ... 1460
31.8.3.2. Open a Connection to the VTR Controller ... 1461
31.8.3.3. Set Up Your Recording Session ... 1461
31.8.3.3.1. Select the Recording Source ... 1463
31.8.3.3.2. Choose Real-Time or Frame-By-Frame Recording ... 1463
31.8.3.3.3. Set the Video Frame Hold Counts ... 1464
31.8.3.4. Check the Picture Quality ... 1464
31.8.3.5. Make Sure Your Tape is Formatted (Preblacked) ... 1465
31.8.3.6. Start the Recording Session ... 1465
31.9. Histogram and XY Plots ... 1466
31.9.1. Plot Types ... 1466
31.9.1.1. XY Plots ... 1466
31.9.1.2. Histograms ... 1467
31.9.2. XY Plots of Solution Data ... 1468
31.9.2.1. Steps for Generating Solution XY Plots ... 1468
31.9.2.2. Options for Solution XY Plots ... 1471
31.9.2.2.1. Including External Data in the Solution XY Plot ... 1472
31.9.2.2.2. Choosing Node or Cell Values ... 1472
31.9.2.2.3. Saving the Plot Data to a File ... 1472
31.9.3. XY Plots of File Data ... 1472
31.9.3.1. Steps for Generating XY Plots of Data in External Files ... 1472
31.9.3.2. Options for File XY Plots ... 1473
31.9.3.2.1. Changing the Plot Title ... 1473
31.9.3.2.2. Changing the Legend Entry ... 1473
31.9.3.2.3. Changing the Legend Title ... 1474
31.9.4. XY Plots of Profiles ... 1474
31.9.4.1. Steps for Generating Plots of Profile Data ... 1474
31.9.4.2. Steps for Generating Plots of Interpolated Profile Data ... 1475
31.9.5. XY Plots of Circumferential Averages ... 1475
31.9.5.1. Steps for Generating an XY Plot of Circumferential Averages ... 1476
31.9.5.2. Customizing the Appearance of the Plot ... 1477
31.9.6. XY Plot File Format ... 1477
31.9.7. Residual Plots ... 1478
31.9.8. Histograms ... 1478
31.9.8.1. Steps for Generating Histogram Plots ... 1478
31.9.8.2. Options for Histogram Plots ... 1479
31.9.8.2.1. Specifying the Range of Values Plotted ... 1480
31.9.9. Modifying Axis Attributes ... 1480
31.9.9.1. Using the Axes Dialog Box ... 1480
31.9.9.1.1. Changing the Axis Label ... 1481
31.9.9.1.2. Changing the Format of the Data Labels ... 1481
31.9.9.1.3. Choosing Logarithmic or Decimal Scaling ... 1481
31.9.9.1.4. Resetting the Range of the Axis ... 1481
31.9.9.1.5. Controlling the Major and Minor Rules ... 1481
31.9.10. Modifying Curve Attributes ... 1482
31.9.10.1. Using the Curves Dialog Box ... 1482
31.9.10.1.1. Changing the Line Style ... 1482
31.9.10.1.2. Changing the Marker Style ... 1483
31.9.10.1.3. Previewing the Curve Style ... 1483
31.10. Turbomachinery Postprocessing ... 1483
31.10.1. Defining the Turbomachinery Topology ... 1484

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.1.1. Boundary Types ... 1486
31.10.2. Generating Reports of Turbomachinery Data ... 1487
31.10.2.1. Computing Turbomachinery Quantities ... 1488
31.10.2.1.1. Mass Flow ... 1488
31.10.2.1.2. Swirl Number ... 1488
31.10.2.1.3. Average Total Pressure ... 1488
31.10.2.1.4. Average Total Temperature ... 1489
31.10.2.1.5. Average Flow Angles ... 1489
31.10.2.1.6. Passage Loss Coefficient ... 1490
31.10.2.1.7. Axial Force ... 1491
31.10.2.1.8. Torque ... 1491
31.10.2.1.9. Efficiencies for Pumps and Compressors ... 1491
31.10.2.1.9.1. Incompressible Flows ... 1492
31.10.2.1.9.2. Compressible Flows ... 1492
31.10.2.1.10. Efficiencies for Turbines ... 1494
31.10.2.1.10.1. Incompressible Flows ... 1494
31.10.2.1.10.2. Compressible Flows ... 1494
31.10.3. Displaying Turbomachinery Averaged Contours ... 1496
31.10.3.1. Steps for Generating Turbomachinery Averaged Contour Plots ... 1496
31.10.3.2. Contour Plot Options ... 1497
31.10.4. Displaying Turbomachinery 2D Contours ... 1497
31.10.4.1. Steps for Generating Turbo 2D Contour Plots ... 1497
31.10.4.2. Contour Plot Options ... 1498
31.10.5. Generating Averaged XY Plots of Turbomachinery Solution Data ... 1499
31.10.5.1. Steps for Generating Turbo Averaged XY Plots ... 1499
31.10.6. Globally Setting the Turbomachinery Topology ... 1500
31.10.7. Turbomachinery-Specific Variables ... 1500
31.11. Fast Fourier Transform (FFT) Postprocessing ... 1500
31.11.1. Limitations of the FFT Algorithm ... 1501
31.11.2. Windowing ... 1501
31.11.3. Fast Fourier Transform (FFT) ... 1502
31.11.4. Using the FFT Utility ... 1503
31.11.4.1. Loading Data for Spectral Analysis ... 1504
31.11.4.2. Customizing the Input ... 1504
31.11.4.2.1. Customizing the Input Signal Data Set ... 1505
31.11.4.2.2. Viewing Data Statistics ... 1505
31.11.4.2.3. Customizing Titles and Labels ... 1505
31.11.4.2.4. Applying the Changes in the Input Signal Data ... 1505
31.11.4.3. Customizing the Output ... 1505
31.11.4.3.1. Specifying a Function for the y Axis ... 1505
31.11.4.3.2. Specifying a Function for the x Axis ... 1507
31.11.4.3.3. Specifying Output Options ... 1508
31.11.4.3.4. Specifying a Windowing Technique ... 1508
31.11.4.3.5. Specifying Labels and Titles ... 1508
32. Reporting Alphanumeric Data ... 1509
32.1. Reporting Conventions ... 1509
32.2. Creating Output Parameters ... 1509
32.3. Fluxes Through Boundaries ... 1510
32.3.1. Generating a Flux Report ... 1511
32.3.2. Flux Reporting for Reacting Flows ... 1512
32.3.2.1. Flux Reporting with Particles ... 1514
32.3.2.2. Flux Reporting with Multiphase ... 1514

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.3.2.3. Flux Reporting with Other Volumetric Sources ... 1515
32.4. Forces on Boundaries ... 1515
32.4.1. Generating a Force, Moment, or Center of Pressure Report ... 1515
32.4.1.1. Example ... 1517
32.5. Projected Surface Area Calculations ... 1518
32.6. Surface Integration ... 1519
32.6.1. Generating a Surface Integral Report ... 1519
32.7. Volume Integration ... 1521
32.7.1. Generating a Volume Integral Report ... 1521
32.8. Histogram Reports ... 1522
32.9. Discrete Phase ... 1523
32.10. S2S Information ... 1523
32.11. Reference Values ... 1523
32.11.1. Setting Reference Values ... 1524
32.11.2. Setting the Reference Zone ... 1525
32.12. Summary Reports of Case Settings ... 1525
32.12.1. Generating a Summary Report ... 1525
32.13. Memory and CPU Usage ... 1526
33. Field Function Definitions ... 1529
33.1. Node, Cell, and Facet Values ... 1529
33.1.1. Cell Values ... 1529
33.1.2. Node Values ... 1529
33.1.2.1. Vertex Values for Points That are Not Mesh Nodes ... 1530
33.1.3. Facet Values ... 1530
33.1.3.1. Facet Values on Zone Surfaces ... 1530
33.1.3.2. Facet Values on Postprocessing Surfaces ... 1531
33.2. Velocity Reporting Options ... 1531
33.3. Field Variables Listed by Category ... 1532
33.4. Alphabetical Listing of Field Variables and Their Definitions ... 1547
33.5. Custom Field Functions ... 1579
33.5.1. Creating a Custom Field Function ... 1580
33.5.1.1. Using the Calculator Buttons ... 1582
33.5.1.2. Using the Field Functions List ... 1582
33.5.2. Manipulating, Saving, and Loading Custom Field Functions ... 1582
33.5.3. Sample Custom Field Functions ... 1583
34. Parallel Processing ... 1585
34.1. Introduction to Parallel Processing ... 1585
34.1.1. Recommended Usage of Parallel ANSYS FLUENT ... 1587
34.2. Starting Parallel ANSYS FLUENT Using FLUENT Launcher ... 1587
34.2.1. Setting Parallel Scheduler Options in FLUENT Launcher ... 1589
34.2.2. Setting Additional Options When Running on Remote Linux Machines ... 1591
34.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines ... 1593
34.3. Starting Parallel ANSYS FLUENT on a Windows System ... 1594
34.3.1. Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options ... 1595
34.3.1.1. Starting Parallel ANSYS FLUENT with the Microsoft Job Scheduler ... 1597
34.4. Starting Parallel ANSYS FLUENT on a Linux/UNIX System ... 1599
34.4.1. Starting Parallel ANSYS FLUENT on a Linux/UNIX System Using Command Line Options ... 1599
34.4.2. Setting Up Your Remote Shell and Secure Shell Clients ... 1602
34.4.2.1. Configuring the `rsh` Client ... 1602
34.4.2.2. Configuring the `ssh` Client ... 1602
34.5. Mesh Partitioning and Load Balancing ... 1603
34.5.1. Overview of Mesh Partitioning ... 1603

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.5.2. Partitioning the Mesh Automatically ... 1604
34.5.2.1. Reporting During Auto Partitioning ... 1605
34.5.3. Partitioning the Mesh Manually and Balancing the Load ... 1605
34.5.3.1. Guidelines for Partitioning the Mesh ... 1605
34.5.4. Using the Partitioning and Load Balancing Dialog Box ... 1606
34.5.4.1. Partitioning ... 1606
34.5.4.1.1. Example of Setting Selected Registers to Specified Partition IDs ... 1611
34.5.4.1.2. Partitioning Within Zones or Registers ... 1613
34.5.4.1.3. Reporting During Partitioning ... 1613
34.5.4.1.4. Resetting the Partition Parameters ... 1614
34.5.4.2. Load Balancing ... 1614
34.5.5. Mesh Partitioning Methods ... 1616
34.5.5.1. Partition Methods ... 1616
34.5.5.2. Optimizations ... 1620
34.5.5.3. Pretesting ... 1621
34.5.5.4. Using the Partition Filter ... 1622
34.5.6. Checking the Partitions ... 1622
34.5.6.1. Interpreting Partition Statistics ... 1622
34.5.6.2. Examining Partitions Graphically ... 1623
34.5.7. Load Distribution ... 1624
34.5.8. Troubleshooting ... 1624
34.6. Controlling the Threads ... 1625
34.7. Checking Network Connectivity ... 1625
34.8. Checking and Improving Parallel Performance ... 1626
34.8.1. Checking Parallel Performance ... 1626
34.8.1.1. Checking Latency and Bandwidth ... 1628
34.8.2. Optimizing the Parallel Solver ... 1630
34.8.2.1. Increasing the Report Interval ... 1630
35. Task Page Reference Guide ... 1631
35.1. Problem Setup Task Page ... 1631
35.2. General Task Page ... 1631
35.2.1. Scale Mesh Dialog Box ... 1634
35.2.2. Mesh Display Dialog Box ... 1635
35.2.3. Set Units Dialog Box ... 1637
35.2.4. Define Unit Dialog Box ... 1638
35.2.5. Mesh Colors Dialog Box ... 1638
35.3. Models Task Page ... 1639
35.3.1. Multiphase Model Dialog Box ... 1642
35.3.2. Energy Dialog Box ... 1645
35.3.3. Viscous Model Dialog Box ... 1645
35.3.4. Radiation Model Dialog Box ... 1657
35.3.5. View Factors and Clustering Dialog Box ... 1660
35.3.6. Participating Boundary Zones Dialog Box ... 1663
35.3.7. Solar Calculator Dialog Box ... 1665
35.3.8. Heat Exchanger Model Dialog Box ... 1666
35.3.9. Dual Cell Heat Exchanger Dialog Box ... 1667
35.3.10. Set Dual Cell Heat Exchanger Dialog Box ... 1668
35.3.11. Heat Transfer Data Table Dialog Box ... 1670
35.3.12. NTU Table Dialog Box ... 1671
35.3.13. Copy From Dialog Box ... 1672
35.3.14. Ungrouped Macro Heat Exchanger Dialog Box ... 1672
35.3.15. Velocity Effectiveness Curve Dialog Box ... 1675

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3.16. Core Porosity Model Dialog Box 1676
35.3.17. Macro Heat Exchanger Group Dialog Box 1677
35.3.18. Species Model Dialog Box 1681
35.3.19. Coal Calculator Dialog Box 1693
35.3.20. Integration Parameters Dialog Box 1694
35.3.21. Chemkin Mechanism Import Dialog Box 1696
35.3.22. Flamelet 3D Surfaces Dialog Box 1697
35.3.23. Flamelet 2D Curves Dialog Box 1699
35.3.24. Spark Ignition Dialog Box 1700
35.3.25. Set Spark Ignition Dialog Box 1700
35.3.26. Autoignition Model Dialog Box 1703
35.3.27. Inert Dialog Box 1705
35.3.28. NOx Model Dialog Box 1706
35.3.29. SOx Model Dialog Box 1713
35.3.30. Soot Model Dialog Box 1717
35.3.31. Decoupled Detailed Chemistry Dialog Box 1722
35.3.32. Discrete Phase Model Dialog Box 1723
35.3.33. Solidification and Melting Dialog Box 1730
35.3.34. Acoustics Model Dialog Box 1731
35.3.35. Acoustic Sources Dialog Box 1733
35.3.36. Acoustic Receivers Dialog Box 1734
35.3.37. Interior Cell Zone Selection Dialog Box 1734
35.4. Materials Task Page 1735
35.4.1. Create/Edit Materials Dialog Box 1736
35.4.2. FLUENT Database Materials Dialog Box 1744
35.4.3. Open Database Dialog Box 1746
35.4.4. User-Defined Database Materials Dialog Box 1746
35.4.5. Copy Case Material Dialog Box 1748
35.4.6. Material Properties Dialog Box 1748
35.4.7. Edit Property Methods Dialog Box 1749
35.4.8. New Material Name Dialog Box 1750
35.4.9. Polynomial Profile Dialog Box 1751
35.4.10. Piecewise-Linear Profile Dialog Box 1751
35.4.11. Piecewise-Polynomial Profile Dialog Box 1752
35.4.12. User-Defined Functions Dialog Box 1753
35.4.13. Sutherland Law Dialog Box 1754
35.4.14. Power Law Dialog Box 1755
35.4.15. Non-Newtonian Power Law Dialog Box 1755
35.4.16. Carreau Model Dialog Box 1756
35.4.17. Cross Model Dialog Box 1757
35.4.18. Herschel-Bulkley Dialog Box 1758
35.4.19. Biaxial Conductivity Dialog Box 1759
35.4.20. Cylindrical Orthotropic Conductivity Dialog Box 1760
35.4.21. Orthotropic Conductivity Dialog Box 1761
35.4.22. Anisotropic Conductivity Dialog Box 1762
35.4.23. Species Dialog Box 1763
35.4.24. Reactions Dialog Box 1765
35.4.25. Third-Body Efficiencies Dialog Box 1768
35.4.26. Pressure-Dependent Reaction Dialog Box 1768
35.4.27. Coverage-Dependent Reaction Dialog Box 1770
35.4.28. Reaction Mechanisms Dialog Box 1771
35.4.29. Site Parameters Dialog Box 1772

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.30. Mass Diffusion Coefficients Dialog Box 1773
35.4.31. Thermal Diffusion Coefficients Dialog Box 1774
35.4.32. UDS Diffusion Coefficients Dialog Box 1775
35.4.33. WSGGM User Specified Dialog Box 1776
35.4.34. Gray-Band Absorption Coefficient Dialog Box 1776
35.4.35. Delta-Eddington Scattering Function Dialog Box 1777
35.4.36. Gray-Band Refractive Index Dialog Box 1777
35.4.37. Single Rate Devolatilization Dialog Box 1778
35.4.38. Two Competing Rates Model Dialog Box 1779
35.4.39. CPD Model Dialog Box 1780
35.4.40. Kinetics/Diffusion-Limited Combustion Model Dialog Box 1780
35.4.41. Intrinsic Combustion Model Dialog Box 1781
35.4.42. Edit Material Dialog Box 1782
35.4.43. Fluent Database Materials Dialog Box 1783
35.5. Phases Task Page 1783
35.5.1. Primary Phase Dialog Box 1784
35.5.2. Secondary Phase Dialog Box 1785
35.5.3. Discrete Phase Dialog Box 1789
35.5.4. Phase Interaction Dialog Box 1791
35.6. Cell Zone Conditions Task Page 1794
35.6.1. Fluid Dialog Box 1796
35.6.2. Solid Dialog Box 1802
35.6.3. Copy Conditions Dialog Box 1805
35.6.4. Operating Conditions Dialog Box 1806
35.6.5. Select Input Parameter Dialog Box 1807
35.6.6. Profiles Dialog Box 1808
35.6.7. Orient Profile Dialog Box 1810
35.6.8. Write Profile Dialog Box 1811
35.7. Boundary Conditions Task Page 1812
35.7.1. Axis Dialog Box 1814
35.7.2. Exhaust Fan Dialog Box 1815
35.7.3. Fan Dialog Box 1819
35.7.4. Inlet Vent Dialog Box 1821
35.7.5. Intake Fan Dialog Box 1826
35.7.6. Interface Dialog Box 1831
35.7.7. Interior Dialog Box 1831
35.7.8. Mass-Flow Inlet Dialog Box 1832
35.7.9. Outflow Dialog Box 1836
35.7.10. Outlet Vent Dialog Box 1838
35.7.11. Periodic Dialog Box 1842
35.7.12. Porous Jump Dialog Box 1842
35.7.13. Pressure Far-Field Dialog Box 1844
35.7.14. Pressure Inlet Dialog Box 1848
35.7.15. Pressure Outlet Dialog Box 1853
35.7.16. Radiator Dialog Box 1857
35.7.17. RANS/LES Interface Dialog Box 1858
35.7.18. Symmetry Dialog Box 1859
35.7.19. Velocity Inlet Dialog Box 1860
35.7.20. Wall Dialog Box 1865
35.7.21. Periodic Conditions Dialog Box 1874
35.8. Mesh Interfaces Task Page 1875
35.8.1. Create/Edit Mesh Interfaces Dialog Box 1876

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.9. Dynamic Mesh Task Page ... 1877
35.9.1. Mesh Method Settings Dialog Box ... 1879
35.9.2. Mesh Scale Info Dialog Box ... 1882
35.9.3. Options Dialog Box ... 1883
35.9.4. In-Cylinder Output Controls Dialog Box ... 1884
35.9.5. Dynamic Mesh Events Dialog Box ... 1886
35.9.6. Define Event Dialog Box ... 1887
35.9.7. Events Preview Dialog Box ... 1889
35.9.8. Dynamic Mesh Zones Dialog Box ... 1889
35.9.9. CutCell Boundary Zones Info Dialog Box ... 1894
35.9.10. Zone Scale Info Dialog Box ... 1894
35.9.11. Zone Motion Dialog Box ... 1895
35.9.12. Mesh Motion Dialog Box ... 1896
35.9.13. Autosave Case During Mesh Motion Preview Dialog Box ... 1897
35.10. Reference Values Task Page ... 1898
35.11. Solution Task Page ... 1899
35.12. Solution Methods Task Page ... 1900
35.13. Solution Controls Task Page ... 1902
35.13.1. Equations Dialog Box ... 1904
35.13.2. Solution Limits Dialog Box ... 1904
35.13.3. Advanced Solution Controls Dialog Box ... 1906
35.14. Monitors Task Page ... 1913
35.14.1. Residual Monitors Dialog Box ... 1915
35.14.2. Statistic Monitors Dialog Box ... 1917
35.14.3. Drag Monitor Dialog Box ... 1918
35.14.4. Lift Monitor Dialog Box ... 1920
35.14.5. Moment Monitor Dialog Box ... 1921
35.14.6. Surface Monitor Dialog Box ... 1923
35.14.7. Volume Monitor Dialog Box ... 1925
35.14.8. Point Surface Dialog Box ... 1927
35.14.9. Line/Rake Surface Dialog Box ... 1928
35.14.10. Plane Surface Dialog Box ... 1929
35.14.11. Quadric Surface Dialog Box ... 1931
35.14.12. Iso-Surface Dialog Box ... 1933
35.14.13. Iso-Clip Dialog Box ... 1935
35.14.14. Surfaces Dialog Box ... 1936
35.15. Solution Initialization Task Page ... 1937
35.15.1. Patch Dialog Box ... 1939
35.15.2. Hybrid Initialization Dialog Box ... 1941
35.16. Calculation Activities Task Page ... 1942
35.16.1. Autosave Dialog Box ... 1943
35.16.2. Data File Quantities Dialog Box ... 1945
35.16.3. Automatic Export Dialog Box ... 1946
35.16.4. Automatic Particle History Data Export Dialog Box ... 1949
35.16.5. Execute Commands Dialog Box ... 1950
35.16.6. Define Macro Dialog Box ... 1951
35.16.7. Automatic Solution Initialization and Case Modification Dialog Box ... 1952
35.16.8. Solution Animation Dialog Box ... 1953
35.16.9. Animation Sequence Dialog Box ... 1953
35.17. Run Calculation Task Page ... 1955
35.17.1. Solution Steering Dialog Box ... 1958
35.17.2. Case Check Dialog Box ... 1959

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.17.3. Adaptive Time Step Settings Dialog Box ... 1960
35.17.4. Variable Time Step Settings Dialog Box ... 1962
35.17.5. Sampling Options Dialog Box ... 1963
35.17.6. Acoustic Signals Dialog Box ... 1963
35.18. Results Task Page ... 1965
35.19. Graphics and Animations Task Page ... 1965
35.19.1. Contours Dialog Box ... 1967
35.19.2. Profile Options Dialog Box ... 1970
35.19.3. Vectors Dialog Box ... 1970
35.19.4. Vector Options Dialog Box ... 1973
35.19.5. Custom Vectors Dialog Box ... 1973
35.19.6. Vector Definitions Dialog Box ... 1974
35.19.7. Pathlines Dialog Box ... 1975
35.19.8. Path Style Attributes Dialog Box ... 1979
35.19.9. Ribbon Attributes Dialog Box ... 1979
35.19.10. Particle Tracks Dialog Box ... 1980
35.19.11. Reporting Variables Dialog Box ... 1984
35.19.12. Sweep Surface Dialog Box ... 1984
35.19.13. Create Surface Dialog Box ... 1986
35.19.14. Animate Dialog Box ... 1986
35.19.15. Save Picture Dialog Box ... 1988
35.19.16. Playback Dialog Box ... 1990
35.19.17. Display Options Dialog Box ... 1992
35.19.18. Scene Description Dialog Box ... 1994
35.19.19. Display Properties Dialog Box ... 1996
35.19.20. Transformations Dialog Box ... 1997
35.19.21. Iso-Value Dialog Box ... 1998
35.19.22. Pathline Attributes Dialog Box ... 1999
35.19.23. Bounding Frame Dialog Box ... 2000
35.19.24. Views Dialog Box ... 2000
35.19.25. Write Views Dialog Box ... 2002
35.19.26. Mirror Planes Dialog Box ... 2002
35.19.27. Graphics Periodicity Dialog Box ... 2003
35.19.28. Camera Parameters Dialog Box ... 2005
35.19.29. Lights Dialog Box ... 2005
35.19.30. Colormap Dialog Box ... 2007
35.19.31. Colormap Editor Dialog Box ... 2008
35.19.32. Annotate Dialog Box ... 2010
35.20. Plots Task Page ... 2011
35.20.1. Solution XY Plot Dialog Box ... 2012
35.20.2. Histogram Dialog Box ... 2015
35.20.3. File XY Plot Dialog Box ... 2016
35.20.4. Plot Profile Data Dialog Box ... 2017
35.20.5. Plot Interpolated Data Dialog Box ... 2018
35.20.6. Fourier Transform Dialog Box ... 2019
35.20.7. Plot/Modify Input Signal Dialog Box ... 2020
35.20.8. Axes Dialog Box ... 2022
35.20.9. Curves Dialog Box ... 2024
35.21. Reports Task Page ... 2026
35.21.1. Flux Reports Dialog Box ... 2027
35.21.2. Force Reports Dialog Box ... 2029
35.21.3. Projected Surface Areas Dialog Box ... 2030

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.21.4. Surface Integrals Dialog Box ... 2031
35.21.5. Volume Integrals Dialog Box ... 2034
35.21.6. Sample Trajectories Dialog Box ... 2036
35.21.7. Trajectory Sample Histograms Dialog Box ... 2038
35.21.8. Particle Summary Dialog Box ... 2039
35.21.9. Heat Exchanger Report Dialog Box ... 2040
35.21.10. Parameters Dialog Box ... 2041
35.21.11. Rename Dialog Box ... 2043
35.21.12. Input Parameter Properties Dialog Box ... 2043
35.21.13. Save Output Parameter Dialog Box ... 2044
36. Menu Reference Guide ... 2045
36.1. File Menu ... 2045
36.1.1. File/Read/Mesh... ... 2046
36.1.1.1. Read Mesh Options Dialog Box ... 2047
36.1.2. File/Read/Case... ... 2047
36.1.3. File/Read/Data... ... 2048
36.1.4. File/Read/Case & Data... ... 2048
36.1.5. File/Read/PDF... ... 2048
36.1.6. File/Read/ISAT Table... ... 2049
36.1.7. File/Read/DTRM Rays... ... 2049
36.1.8. File/Read/View Factors... ... 2049
36.1.9. File/Read/Profile... ... 2049
36.1.10. File/Read/Scheme... ... 2049
36.1.11. File/Read/Journal... ... 2049
36.1.12. File/Write/Case... ... 2049
36.1.13. File/Write/Data... ... 2050
36.1.14. File/Write/Case & Data... ... 2050
36.1.15. File/Write/PDF... ... 2050
36.1.16. File/Write/ISAT Table... ... 2050
36.1.17. File/Write/Flamelet... ... 2050
36.1.18. File/Write/Surface Clusters... ... 2050
36.1.19. File/Write/Profile... ... 2051
36.1.20. File/Write/Autosave... ... 2051
36.1.21. File/Write/Boundary Mesh... ... 2051
36.1.22. File/Write/Start Journal... ... 2051
36.1.23. File/Write/Stop Journal ... 2051
36.1.24. File/Write/Start Transcript... ... 2051
36.1.25. File/Write/Stop Transcript ... 2051
36.1.26. File/Import/ABAQUS/Input File... ... 2051
36.1.27. File/Import/ABAQUS/Filbin File... ... 2051
36.1.28. File/Import/ABAQUS/ODB File... ... 2052
36.1.29. File/Import/CFX/Definition File... ... 2052
36.1.30. File/Import/CFX/Result File... ... 2052
36.1.31. File/Import/CGNS/Mesh... ... 2052
36.1.32. File/Import/CGNS/Data... ... 2052
36.1.33. File/Import/CGNS/Mesh & Data... ... 2052
36.1.34. File/Import/EnSight... ... 2052
36.1.35. File/Import/FIDAP... ... 2052
36.1.36. File/Import/GAMBIT... ... 2052
36.1.37. File/Import/HYPERMESH ASCII... ... 2052
36.1.38. File/Import/IC3M... ... 2053
36.1.39. File/Import/I-deas Universal... ... 2053

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.40. File/Import/LSTC/Input File... 2053
36.1.41. File/Import/LSTC/State File... 2053
36.1.42. File/Import/Marc POST... 2053
36.1.43. File/Import/Mechanical APDL/Input File... 2053
36.1.44. File/Import/Mechanical APDL/Result File... 2053
36.1.45. File/Import/NASTRAN/Bulkdata File... 2053
36.1.46. File/Import/NASTRAN/Op2 File... 2054
36.1.47. File/Import/PATRAN/Neutral File... 2054
36.1.48. File/Import/PATRAN/Result File... 2054
36.1.49. File/Import/PLOT3D/Grid File... 2054
36.1.50. File/Import/PLOT3D/Result File... 2054
36.1.51. File/Import/PTC Mechanica Design... 2054
36.1.52. File/Import/Tecplot... 2054
36.1.53. File/Import/FLUENT 4 Case File... 2054
36.1.54. File/Import/PreBFC File... 2054
36.1.55. File/Import/Partition/Metis... 2055
36.1.56. File/Import/Partition/Metis Zone... 2055
36.1.57. File/Import/CHEMKIN Mechanism... 2055
36.1.57.1. CHEMKIN Mechanism Import Dialog Box 2055
36.1.58. File/Export/Solution Data.. 2056
36.1.58.1. Export Dialog Box 2056
36.1.59. File/Export/Particle History Data.. 2060
36.1.59.1. Export Particle History Data Dialog Box 2060
36.1.60. File/Export/During Calculation/Solution Data.. 2061
36.1.61. File/Export/During Calculation/Particle History Data.. 2061
36.1.62. File/Export to CFD-Post... 2061
36.1.62.1. Export to CFD-Post Dialog Box 2061
36.1.63. File/Solution Files... 2062
36.1.63.1. Solution Files Dialog Box 2062
36.1.64. File/Interpolate... 2063
36.1.64.1. Interpolate Data Dialog Box 2063
36.1.65. File/EM Mapping/HFSS... 2064
36.1.65.1. HFSS Mapping Dialog Box 2065
36.1.66. File/EM Mapping/Maxwell... 2065
36.1.66.1. Maxwell Mapping Dialog Box 2066
36.1.67. File/EM Mapping/Q3D... 2067
36.1.67.1. Q3D Mapping Dialog Box 2067
36.1.68. File/FSI Mapping/Volume... 2068
36.1.68.1. Volume FSI Mapping Dialog Box 2068
36.1.69. File/FSI Mapping/Surface... 2070
36.1.69.1. Surface FSI Mapping Dialog Box 2070
36.1.70. File/Save Picture... 2073
36.1.71. File/Data File Quantities... 2073
36.1.72. File/Batch Options... 2073
36.1.72.1. Batch Options Dialog Box 2073
36.1.73. File/RSF... 2074
36.1.74. File/Exit 2074
36.2. Mesh Menu 2074
36.2.1. Mesh/Check 2075
36.2.2. Mesh/Info/Quality 2075
36.2.3. Mesh/Info/Size 2075
36.2.4. Mesh/Info/Memory Usage 2075

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.5. Mesh/Info/Zones ... 2075
36.2.6. Mesh/Info/Partitions ... 2075
36.2.7. Mesh/Polyhedra/Convert Domain ... 2076
36.2.8. Mesh/Polyhedra/Convert Skewed Cells... ... 2076
36.2.8.1. Convert Skewed Cells Dialog Box ... 2076
36.2.9. Mesh/Merge... ... 2077
36.2.9.1. Merge Zones Dialog Box ... 2077
36.2.9.2. Warning Dialog Box ... 2077
36.2.10. Mesh/Separate/Faces... ... 2078
36.2.10.1. Separate Face Zones Dialog Box ... 2078
36.2.11. Mesh/Separate/Cells... ... 2079
36.2.11.1. Separate Cell Zones Dialog Box ... 2080
36.2.12. Mesh/Fuse... ... 2080
36.2.12.1. Fuse Face Zones Dialog Box ... 2080
36.2.13. Mesh/Zone/Append Case File... ... 2081
36.2.14. Mesh/Zone/Append Case & Data Files... ... 2081
36.2.15. Mesh/Zone/Replace... ... 2081
36.2.15.1. Replace Cell Zone Dialog Box ... 2081
36.2.16. Mesh/Zone/Delete... ... 2082
36.2.16.1. Delete Cell Zones Dialog Box ... 2082
36.2.17. Mesh/Zone/Deactivate... ... 2083
36.2.17.1. Deactivate Cell Zones Dialog Box ... 2083
36.2.18. Mesh/Zone/Activate... ... 2083
36.2.18.1. Activate Cell Zones Dialog Box ... 2083
36.2.19. Mesh/Replace... ... 2084
36.2.20. Mesh/Reorder/Domain ... 2084
36.2.21. Mesh/Reorder/Zones ... 2084
36.2.22. Mesh/Reorder/Print Bandwidth ... 2084
36.2.23. Mesh/Scale... ... 2084
36.2.24. Mesh/Translate... ... 2084
36.2.24.1. Translate Mesh Dialog Box ... 2084
36.2.25. Mesh/Rotate... ... 2085
36.2.25.1. Rotate Mesh Dialog Box ... 2085
36.2.26. Mesh/Smooth/Swap... ... 2086
36.3. Define Menu ... 2086
36.3.1. Define/General... ... 2087
36.3.2. Define/Models... ... 2087
36.3.3. Define/Materials... ... 2087
36.3.4. Define/Phases... ... 2087
36.3.5. Define/Cell Zone Conditions... ... 2087
36.3.6. Define/Boundary Conditions... ... 2087
36.3.7. Define/Operating Conditions... ... 2087
36.3.8. Define/Mesh Interfaces... ... 2088
36.3.9. Define/Dynamic Mesh... ... 2088
36.3.10. Define/Mesh Morpher/Optimizer... ... 2088
36.3.10.1. Mesh Morpher/Optimizer Dialog Box ... 2088
36.3.11. Define/Mixing Planes... ... 2093
36.3.11.1. Mixing Planes Dialog Box ... 2093
36.3.12. Define/Turbo Topology... ... 2094
36.3.12.1. Turbo Topology Dialog Box ... 2094
36.3.13. Define/Injections... ... 2096
36.3.13.1. Injections Dialog Box ... 2096

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.13.2. Set Injection Properties Dialog Box ... 2097
36.3.13.3. Set Multiple Injection Properties Dialog Box ... 2101
36.3.13.4. Custom Laws Dialog Box ... 2102
36.3.14. Define/DTRM Rays... ... 2103
36.3.14.1. DTRM Rays Dialog Box ... 2103
36.3.15. Define/Shell Conduction Walls... ... 2104
36.3.15.1. Shell Conduction Walls Dialog Box ... 2104
36.3.15.2. Set Shell Thickness Dialog Box ... 2105
36.3.16. Define/Custom Field Functions... ... 2105
36.3.16.1. Custom Field Function Calculator Dialog Box ... 2106
36.3.16.2. Field Function Definitions Dialog Box ... 2107
36.3.17. Define/Parameters... ... 2108
36.3.18. Define/Profiles... ... 2108
36.3.19. Define/Units... ... 2108
36.3.20. Define/User-Defined/Functions/Interpreted... ... 2108
36.3.20.1. Interpreted UDFs Dialog Box ... 2108
36.3.21. Define/User-Defined/Functions/Compiled... ... 2109
36.3.21.1. Compiled UDFs Dialog Box ... 2109
36.3.22. Define/User-Defined/Functions/Manage... ... 2109
36.3.22.1. UDF Library Manager Dialog Box ... 2110
36.3.23. Define/User-Defined/Function Hooks... ... 2110
36.3.23.1. User-Defined Function Hooks Dialog Box ... 2110
36.3.24. Define/User-Defined/Execute on Demand... ... 2113
36.3.24.1. Execute on Demand Dialog Box ... 2113
36.3.25. Define/User-Defined/Scalars... ... 2113
36.3.25.1. User-Defined Scalars Dialog Box ... 2113
36.3.26. Define/User-Defined/Memory... ... 2114
36.3.26.1. User-Defined Memory Dialog Box ... 2114
36.3.27. Define/User-Defined/Fan Model... ... 2115
36.3.27.1. User-Defined Fan Model Dialog Box ... 2115
36.3.28. Define/User-Defined/1D Coupling... ... 2116
36.3.28.1. 1D Simulation Library Dialog Box ... 2116
36.4. Solve Menu ... 2116
36.4.1. Solve/Methods... ... 2116
36.4.2. Solve/Controls... ... 2117
36.4.3. Solve/Monitors... ... 2117
36.4.4. Solve/Initialization... ... 2117
36.4.5. Solve/Calculation Activities... ... 2117
36.4.6. Solve/Run Calculation.... ... 2117
36.5. Adapt Menu ... 2117
36.5.1. Adapt/Boundary... ... 2117
36.5.1.1. Boundary Adaption Dialog Box ... 2117
36.5.2. Adapt/Gradient... ... 2119
36.5.2.1. Gradient Adaption Dialog Box ... 2119
36.5.3. Adapt/Iso-Value... ... 2121
36.5.3.1. Iso-Value Adaption Dialog Box ... 2121
36.5.4. Adapt/Region... ... 2122
36.5.4.1. Region Adaption Dialog Box ... 2122
36.5.5. Adapt/Volume... ... 2124
36.5.5.1. Volume Adaption Dialog Box ... 2124
36.5.6. Adapt/Yplus/Ystar... ... 2126
36.5.6.1. Yplus/Ystar Adaption Dialog Box ... 2126

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.7. Adapt/Anisotropic... 2127
36.5.7.1. Anisotropic Adaption Dialog Box 2127
36.5.8. Adapt/Manage... 2128
36.5.8.1. Manage Adaption Registers Dialog Box 2129
36.5.9. Adapt/Controls... 2130
36.5.9.1. Mesh Adaption Controls Dialog Box 2130
36.5.10. Adapt/Geometry... 2131
36.5.10.1. Geometry Based Adaption Dialog Box 2132
36.5.10.2. Surface Meshes Dialog Box 2132
36.5.10.3. Geometry Based Adaption Controls Dialog Box 2133
36.5.11. Adapt/Display Options... 2134
36.5.11.1. Adaption Display Options Dialog Box 2134
36.5.12. Adapt/Smooth/Swap... 2135
36.5.12.1. Smooth/Swap Mesh Dialog Box 2135
36.6. Surface Menu 2136
36.6.1. Surface/Zone... 2137
36.6.1.1. Zone Surface Dialog Box 2137
36.6.2. Surface/Partition... 2137
36.6.2.1. Partition Surface Dialog Box 2137
36.6.3. Surface/Point... 2139
36.6.4. Surface/Line/Rake... 2139
36.6.5. Surface/Plane... 2139
36.6.6. Surface/Quadric... 2139
36.6.7. Surface/Iso-Surface... 2139
36.6.8. Surface/Iso-Clip... 2139
36.6.9. Surface/Transform... 2139
36.6.9.1. Transform Surface Dialog Box 2139
36.6.10. Surface/Manage... 2141
36.7. Display Menu 2141
36.7.1. Display/Mesh... 2141
36.7.2. Display/Graphics and Animations... 2141
36.7.3. Display/Plots... 2141
36.7.4. Display/Residuals... 2141
36.7.5. Display/Options... 2142
36.7.6. Display/Scene... 2142
36.7.7. Display/Views... 2142
36.7.8. Display/Lights... 2142
36.7.9. Display/Colormap... 2142
36.7.10. Display/Annotate... 2142
36.7.11. Display/Zone Motion... 2142
36.7.12. Display/DTRM Graphics... 2142
36.7.12.1. DTRM Graphics Dialog Box 2142
36.7.13. Display/Import Particle Data... 2144
36.7.13.1. Import Particle Data Dialog Box 2144
36.7.14. Display/PDF Tables/Curves... 2145
36.7.14.1. PDF Table Dialog Box 2145
36.7.15. Display/Video Control... 2148
36.7.15.1. Video Control Dialog Box 2148
36.7.15.2. V-LAN Settings Dialog Box 2150
36.7.15.3. MiniVAS Settings Dialog Box 2152
36.7.15.4. Animation Recording Options Dialog Box 2153
36.7.15.5. Picture Options Dialog Box 2155

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.7.16. Display/Mouse Buttons... 2157
36.7.16.1. Mouse Buttons Dialog Box 2157
36.8. Report Menu 2158
36.8.1. Report/Result Reports... 2158
36.8.2. Report/Input Summary... 2158
36.8.2.1. Input Summary Dialog Box 2158
36.8.3. Report/S2S Information... 2159
36.8.3.1. S2S Information Dialog Box 2159
36.8.4. Report/Reference Values... 2160
36.9. Parallel Menu 2160
36.9.1. Parallel/Auto Partition... 2160
36.9.1.1. Auto Partition Mesh Dialog Box 2160
36.9.2. Parallel/Partitioning and Load Balancing... 2161
36.9.2.1. Partitioning and Load Balancing Dialog Box 2161
36.9.3. Parallel/Thread Control... 2165
36.9.3.1. Thread Control Dialog Box 2165
36.9.4. Parallel/Network/Database... 2166
36.9.4.1. Hosts Database Dialog Box 2166
36.9.5. Parallel/Network/Configure... 2167
36.9.5.1. Network Configuration Dialog Box 2168
36.9.6. Parallel/Network/Show Connectivity... 2170
36.9.6.1. Parallel Connectivity Dialog Box 2171
36.9.7. Parallel/Network/Show Latency 2171
36.9.8. Parallel/Network/Show Bandwidth 2171
36.9.9. Parallel/Timer/Usage 2171
36.9.10. Parallel/Timer/Reset 2171
36.10. View Menu 2171
36.10.1. View/Toolbars 2172
36.10.2. View/Navigation Pane 2172
36.10.3. View/Task Page 2172
36.10.4. View/Graphics Window 2172
36.10.5. View/Embed Graphics Window 2172
36.10.6. View/Show All 2172
36.10.7. View/Show Only Console 2172
36.10.8. View/Graphics Window Layout 2172
36.10.9. View/Save Layout 2172
36.11. Turbo Menu 2172
36.11.1. Turbo/Report... 2173
36.11.1.1. Turbo Report Dialog Box 2173
36.11.2. Turbo/Averaged Contours... 2175
36.11.2.1. Turbo Averaged Contours Dialog Box 2175
36.11.3. Turbo/2D Contours... 2176
36.11.3.1. Turbo 2D Contours Dialog Box 2176
36.11.4. Turbo/Averaged XY Plot... 2178
36.11.4.1. Turbo Averaged XY Plot Dialog Box 2178
36.11.5. Turbo/Options... 2179
36.11.5.1. Turbo Options Dialog Box 2179
36.12. Help Menu 2179
36.12.1. Help/User's Guide Contents... 2180
36.12.2. Help/User's Guide Index... 2180
36.12.3. Help/PDF... 2180
36.12.4. Help/Context-Sensitive Help 2180

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.12.5. Help/Using Help... 2180
36.12.6. Help/Online Technical Resources... 2180
36.12.7. Help/License Usage 2180
36.12.8. Help/Version... 2180
A. ANSYS FLUENT Model Compatibility 2183
B. Case and Data File Formats 2187
B.1. Guidelines 2187
B.2. Formatting Conventions in Binary and Formatted Files 2187
B.3. Grid Sections 2188
B.3.1. Comment 2188
B.3.2. Header 2188
B.3.3. Dimensions 2189
B.3.4. Nodes 2189
B.3.5. Periodic Shadow Faces 2190
B.3.6. Cells 2191
B.3.7. Faces 2192
B.3.8. Face Tree 2194
B.3.9. Cell Tree 2194
B.3.10. Interface Face Parents 2195
B.3.10.1. Example Files 2195
B.3.10.1.1. Example 1 2195
B.3.10.1.2. Example 2 2196
B.3.10.1.3. Example 3 2197
B.4. Other (Non-Grid) Case Sections 2199
B.4.1. Zone 2199
B.4.2. Partitions 2200
B.5. Data Sections 2201
B.5.1. Grid Size 2201
B.5.2. Data Field 2201
B.5.3. Residuals 2202
C. Nomenclature 2205
Bibliography 2209
Index 2217

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Using This Manual

This preface is divided into the following sections:

1. The Contents of This Manual
2. The Contents of the Other Manuals
3. Typographical Conventions
4. Mathematical Conventions
5. Technical Support

1. The Contents of This Manual

The ANSYS FLUENT User's Guide tells you what you need to know to use ANSYS FLUENT. At the end of the User's Guide, you will find a Reference Guide, a nomenclature list, a bibliography, and an index.

Important

Under U.S. and international copyright law, ANSYS, Inc. is unable to distribute copies of the papers listed in the bibliography, other than those published internally by ANSYS, Inc. Please use your library or a document delivery service to obtain copies of copyrighted papers.

A brief description of what is in each chapter follows:

- *Starting and Executing ANSYS FLUENT* (p. 1), describes options and alternatives to starting, running, and exiting ANSYS FLUENT. It also provides instructions for remote execution, batch execution, and using the Remote Simulation Facility (RSF).
- *Graphical User Interface (GUI)* (p. 21), describes the mechanics of using the graphical user interface and the GUI on-line help.
- *Text User Interface (TUI)* (p. 47), describes the mechanics of using the text interface and the TUI on-line help. (See the Text Command List for information about specific text interface commands.)
- *Reading and Writing Files* (p. 57), describes the files that ANSYS FLUENT can read and write, including picture files.
- *Unit Systems* (p. 125), describes how to use the standard and custom unit systems available in ANSYS FLUENT.
- *Reading and Manipulating Meshes* (p. 129), describes the various sources of computational meshes and explains how to obtain diagnostic information about the mesh and how to modify it by scaling, translating, and other methods. This chapter also contains information about the use of non-conformal meshes.
- *Cell Zone and Boundary Conditions* (p. 205), describes the different types of boundary conditions available in ANSYS FLUENT, when to use them, how to define them, and how to define boundary profiles and volumetric sources and fix the value of a variable in a particular region. It also contains information about porous media and lumped parameter models.
- *Physical Properties* (p. 385), describes the physical properties of materials and the equations that ANSYS FLUENT uses to compute the properties from the information that you input.
- *Modeling Basic Fluid Flow* (p. 485), describes the governing equations and physical models used by ANSYS FLUENT to compute fluid flow (including periodic flow, swirling and rotating flows, compressible flows, and inviscid flows), as well as the inputs you need to provide to use these models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *Modeling Flows with Moving Reference Frames* (p. 513), describes the use of single moving reference frames, multiple moving reference frames, and mixing planes in ANSYS FLUENT.
- *Modeling Flows Using Sliding and Dynamic Meshes* (p. 535), describes the use of sliding and deforming meshes in ANSYS FLUENT.
- *Modeling Flows Using the Mesh Morpher/Optimizer* (p. 631), describes ANSYS FLUENT's mesh morphing capability that allows you to solve shape optimization problems [*37*] (p. 2211).
- *Modeling Turbulence* (p. 643), describes ANSYS FLUENT's models for turbulent flow and when and how to use them.
- *Modeling Heat Transfer* (p. 691), describes the physical models used by ANSYS FLUENT to compute heat transfer (including convective and conductive heat transfer, natural convection, radiative heat transfer, and periodic heat transfer), as well as the inputs you need to provide to use these models.
- *Modeling Heat Exchangers* (p. 769), describes ANSYS FLUENT's heat exchanger models and how to use them.
- *Modeling Species Transport and Finite-Rate Chemistry* (p. 799), describes the finite-rate chemistry models in ANSYS FLUENT and how to use them. This chapter also provides information about modeling species transport in non-reacting flows.
- *Modeling Non-Premixed Combustion* (p. 839), describes the non-premixed combustion model and how to use it. This chapter includes details about using prePDF.
- *Modeling Premixed Combustion* (p. 893), describes the premixed combustion model and how to use it.
- *Modeling Partially Premixed Combustion* (p. 905), describes the partially premixed combustion model and how to use it.
- *Modeling a Composition PDF Transport Problem* (p. 911), describes the composition PDF transport model and how to use it.
- *Using Chemistry Acceleration* (p. 923), describes the methods used to accelerate computations for detailed chemical mechanisms involving laminar and turbulent flames.
- *Modeling Engine Ignition* (p. 929), describes the engine ignition models available in ANSYS FLUENT.
- *Modeling Pollutant Formation* (p. 943), describes the models for the formation of NOx, SOx, and soot and how to use them.
- *Predicting Aerodynamically Generated Noise* (p. 987), describes the acoustics model and how to use it.
- *Modeling Discrete Phase* (p. 1007), describes the discrete phase models available in ANSYS FLUENT and how to use them.
- *Modeling Multiphase Flows* (p. 1091), describes the general multiphase models available in ANSYS FLUENT (VOF, mixture, and Eulerian) and how to use them.
- *Modeling Solidification and Melting* (p. 1201), describes ANSYS FLUENT's model for solidification and melting and how to use it.
- *Using the Solver* (p. 1207), describes the ANSYS FLUENT solvers and how to use them.
- *Adapting the Mesh* (p. 1325), describes the solution-adaptive mesh refinement feature in ANSYS FLUENT and how to use it.
- *Creating Surfaces for Displaying and Reporting Data* (p. 1355), describes how to create surfaces in the domain on which you can examine ANSYS FLUENT solution data.
- *Displaying Graphics* (p. 1379), describes the graphics tools that you can use to examine your ANSYS FLUENT solution.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *Reporting Alphanumeric Data* (p. 1509), describes how to obtain reports of fluxes, forces, surface integrals, and other solution data.
- *Field Function Definitions* (p. 1529), describes the flow variables that appear in the variable selection drop-down lists in ANSYS FLUENT dialog boxes, and tells you how to create your own custom field functions.
- *Parallel Processing* (p. 1585), describes the parallel processing features in ANSYS FLUENT and how to use them. This chapter also provides information about partitioning your mesh for parallel processing.
- *Task Page Reference Guide* (p. 1631), describes the task pages within ANSYS FLUENT user interface and how to use them.
- *Menu Reference Guide* (p. 2045), describes the menus within ANSYS FLUENT user interface and how to use them.
- *Appendix A* (p. 2183), presents a chart outlining the compatibility of several ANSYS FLUENT model categories.
- *Appendix B* (p. 2187), presents information about the contents and formats of ANSYS FLUENT case and data files.
- *Appendix C* (p. 2205), presents a brief listing of the physical properties and fluid dynamic quantities used throughout the ANSYS FLUENT documentation.

2. The Contents of the Other Manuals

In addition to this User's Guide, there are several other manuals available to help you use ANSYS FLUENT and its associated programs:

- The Getting Started Guide describes the capabilities of ANSYS FLUENT, provides an overview of the problem setup steps, and presents helpful tips in order for you to create a successful CFD simulation. The manual also includes information about accessing the ANSYS FLUENT manuals in the installation area, and briefly describes the user interface.
- The Tutorial Guide contains a number of example problems with detailed instructions, commentary, and postprocessing of results.
- The UDF Manual contains information about writing and using user-defined functions (UDFs).
- The Text Command List provides a brief description of each of the commands in ANSYS FLUENT's text interface.

3. Typographical Conventions

Several typographical conventions are used in this manual's text to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., **Iso-Surface** dialog box, `surface/iso-surface` command).
- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type into a field in a dialog box. The information displayed on the screen is enclosed in a large box to distinguish it from the narrative text, and user inputs are often enclosed in smaller boxes.
- A mini flow chart is used to guide you through the navigation pane, which leads you to a specific task page or dialog box. For example,

  Models → **Multiphase** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

indicates that **Models** is selected in the navigation pane, which then opens the corresponding task page. In the **Models** task page, **Multiphase** is selected from the list. Clicking the **Edit...** button opens the **Multiphase** dialog box.

Also, a mini flow chart is used to indicate the menu selections that lead you to a specific command or dialog box. For example,

Define $\rightarrow$ **Injections...**

indicates that the **Injections...** menu item can be selected from the **Define** pull-down menu, and

`display` $\rightarrow$ `mesh`

indicates that the `mesh` command is available in the `display` text menu.

In this manual, mini flow charts usually precede a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They allow you to look up information about a command or dialog box and quickly determine how to access it without having to search the preceding material.

- The menu selections that will lead you to a particular dialog box or task page are also indicated (usually within a paragraph) using a " **/**". For example, **Define/Materials...** tells you to choose the **Materials...** menu item from the **Define** pull-down menu.

4. Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (e.g., $\vec{a}$, $\vec{A}$). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (e.g., the identity matrix, I).
- The operator ∇, referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{\imath} + \frac{\partial}{\partial y}\vec{\jmath} + \frac{\partial}{\partial z}\vec{k} \tag{1}$$

∇ appears in several ways:

– The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

$$\nabla p = \frac{\partial p}{\partial x}\vec{\imath} + \frac{\partial p}{\partial y}\vec{\jmath} + \frac{\partial p}{\partial z}\vec{k} \tag{2}$$

– The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v}) = \left(\frac{\partial}{\partial x}\vec{\imath} + \frac{\partial}{\partial y}\vec{\jmath} + \frac{\partial}{\partial z}\vec{k}\right)\left(v_x\vec{\imath} + v_j\vec{\jmath} + v_z\vec{k}\right) \tag{3}$$

This tensor is usually written as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{pmatrix} \frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z} \\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z} \\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z} \end{pmatrix} \tag{4}$$

– The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \tag{5}$$

– The operator $\nabla \cdot \nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \tag{6}$$

$\nabla^2 T$ is different from the expression $(\nabla T)^2$, which is defined as

$$(\nabla T)^2 = \left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2 \tag{7}$$

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in "Modeling Turbulence" in the User's Guide, where convention dictates the use of Cartesian tensor notation. In this chapter, you will also find that some velocity vector components are written as u, v, and w instead of the conventional v with directional subscripts.

5. Technical Support

If you encounter difficulties while using ANSYS FLUENT, please first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal (`www.ansys.com/customerportal`).

If you encounter an error, please write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS FLUENT.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to www.ansys.com and select **Support> Technical Support> Designated Service Providers**. The direct URL is: http://www1.ansys.com/customer/public/supportlist.asp. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: http://www.ansys.com/customerportal.

One of the many useful features of the Customer Portal is the Knowledge Base Search, where you can find solutions to various types of problems, like FAQ. The Knowledge Base Search feature is located under **Online Support> Search Options> Solutions Search**.

Systems and installation Knowledge Resources and FAQs are easily accessible via the Customer Portal under Online Support > Installation/System FAQs. These Knowledge Resources provide a plethora of solutions and direction on how to get installation and licensing issues resolved quickly and efficiently.

NORTH AMERICA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199

Telephone:1.724.514.3600

Fax: 1.724.514.5096

For installation and licensing questions, visit our Knowledge Resources and FAQs on the Customer Portal.

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Fax: +49 (0) 8092 7005-5

Email: support@cadfem.de

CFX Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +49 (0) 8024 9054-44

Fax: +49 (0) 8024 9054-17

Email: cfx-support-germany@ansys.com

FLUENT Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +49 (0) 6151 3644-0

Fax: +49 (0) 6151 3644-44

Email: fluent-support-germany@ansys.com

ICEM CFD Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +49 (0) 511 288696-4

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fax: +49 (0) 511 288696-66

Email: icemcfd-support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email: *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

FLUENT Products

Email: *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

Licensing and Installation

Telephone: +81-3-5324-7305

Email: japan-license-support@ansys.com

INDIA

ANSYS Products (including FLUENT, CFX, ICEM-CFD)

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

Fax: +91 80 2529 1271

Email: *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

FRANCE

ANSYS, CFX, FLUENT, and ICEM CFD Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +33 (0) 820 480 240

Email: *ANSYS:* ansys-support-france@ansys.com; *CFX:* cfx-support-france@ansys.com; *FLUENT:* fluent-support-france@ansys.com; *ICEM CFD:* icemcfd-support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +39 02 89013378

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Starting and Executing ANSYS FLUENT

This chapter provides instructions for starting and executing ANSYS FLUENT.

1.1. Starting ANSYS FLUENT
1.2. Running ANSYS FLUENT in Batch Mode
1.3. Checkpointing an ANSYS FLUENT Simulation
1.4. Cleaning Up Processes From an ANSYS FLUENT Simulation
1.5. Exiting the Program

1.1. Starting ANSYS FLUENT

The way you start ANSYS FLUENT will be the same for Linux/UNIX and Windows systems, as described below in *Starting ANSYS FLUENT Using FLUENT Launcher* (p. 2). The installation process (described in the separate installation instructions) is designed to ensure that the ANSYS FLUENT program is launched when you follow the appropriate instructions.

For additional information, please see the following sections:

1.1.1. Single-Precision and Double-Precision Solvers
1.1.2. Starting ANSYS FLUENT Using FLUENT Launcher
1.1.3. Starting ANSYS FLUENT on a Windows System
1.1.4. Starting ANSYS FLUENT on a Linux/UNIX System
1.1.5. Command Line Startup Options
1.1.6. Remote Simulation Facility (RSF)

1.1.1. Single-Precision and Double-Precision Solvers

Both single-precision and double-precision versions of ANSYS FLUENT are available on all computer platforms. For most cases, the single-precision solver will be sufficiently accurate, but certain types of problems may benefit from the use of a double-precision version. Several examples are listed below:

- If your geometry has features of very disparate length scales (e.g., a very long, thin pipe), single-precision calculations may not be adequate to represent the node coordinates.
- If your geometry involves multiple enclosures connected via small-diameter pipes (e.g., automotive manifolds), mean pressure levels in all but one of the zones can be quite large (since you can set only one global reference pressure location). Double-precision calculations may therefore be necessary to resolve the pressure differences that drive the flow, since these will typically be much smaller than the pressure levels.
- For conjugate problems involving high thermal-conductivity ratios and/or high-aspect-ratio meshes, convergence and/or accuracy may be impaired with the single-precision solver, due to inefficient transfer of boundary information.
- For multiphase problems where the population balance model is used to resolve particle size distributions, which could have statistical moments whose values span many orders of magnitude.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

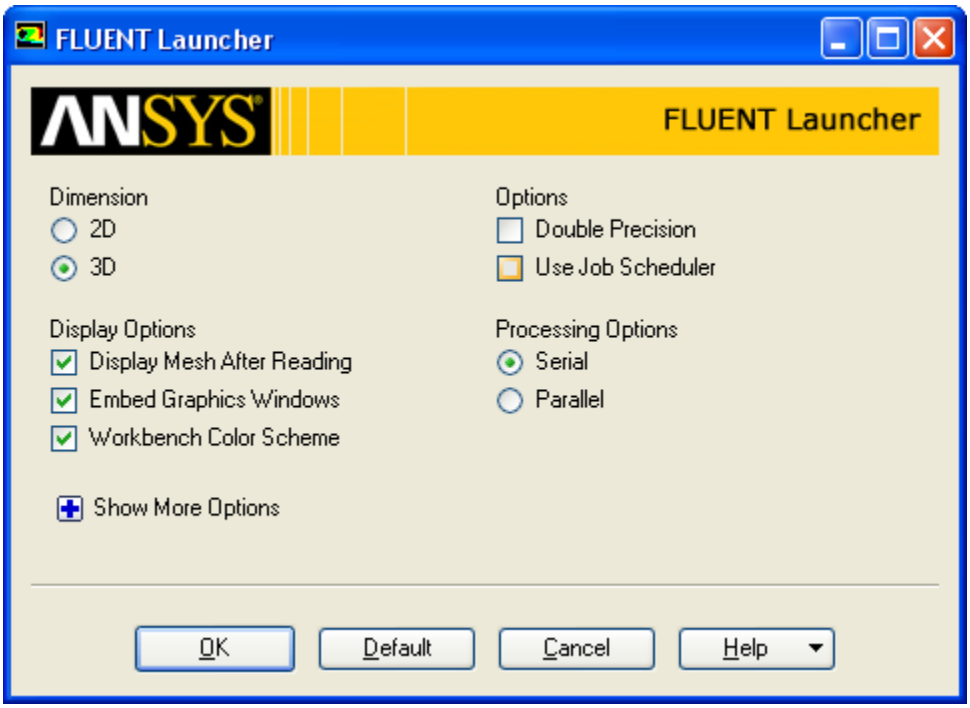
ANSYS FLUENT only allows a period to be used as a decimal separator. If your system is set to a European locale that uses a comma separator (e.g., Germany), fields that accept numeric input may accept a comma, but may ignore everything after the comma. If your system is set to a non-European locale, numeric fields will not accept a comma at all.

ANSYS Workbench accepts commas as decimal delimiters. These are translated into periods when data is passed to ANSYS FLUENT.

1.1.2. Starting ANSYS FLUENT Using FLUENT Launcher

Whether you start ANSYS FLUENT either from the Linux/UNIX or Windows command line with no arguments, from the Windows Programs menu, or from the Windows desktop, FLUENT Launcher will appear, where you can specify the dimensionality of the problem (2D or 3D), as well as other options (e.g., whether you want a single-precision or double-precision calculation):

Figure 1.1 FLUENT Launcher



Under **Dimension**, select **2D** for the two-dimensional solver, or select **3D** for the three-dimensional solver.

The **Display Options** allow you to make decisions related to the graphics windows:

- You can choose to have ANSYS FLUENT automatically display the mesh immediately after reading a mesh or case file by using the **Display Mesh After Reading** option (enabled by default). All of the boundary zones will be displayed, except for the interior zones of 3D geometries. Note that your decision regarding this option can be overridden after you have launched ANSYS FLUENT: simply change the status of the **Display Mesh After Reading** option in the **Select File** dialog box that opens when you are reading in the mesh or case file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can choose to have the graphics windows embedded within the ANSYS FLUENT application window by using the **Embed Graphics Windows** option (enabled by default), rather than having floating graphics windows.
- You can choose to use the default **Workbench Color Scheme** in the graphics windows (i.e., a blue background), rather than the classic black background.

Under **Options**, you can choose to run ANSYS FLUENT in double-precision mode by selecting the **Double-Precision** check box (by default, you start ANSYS FLUENT in single-precision mode). You can also choose to use various job schedulers using the **Use Job Scheduler** option (e.g., the Microsoft Job Scheduler for Windows, or LSF and SGE on Linux/UNIX). For more information about using FLUENT Launcher with job schedulers, see *Setting Scheduler Options in FLUENT Launcher* (p. 7) and *Setting Parallel Scheduler Options in FLUENT Launcher* (p. 1589). In addition, you can also choose to run parallel simulations on Linux clusters, via the Windows interface using the **Use Remote Linux Nodes** option (see *Setting Remote Options in FLUENT Launcher* (p. 6) for details).

Under **Processing Options**, select **Serial** for the serial solver, or select **Parallel** to run the solver in parallel (see *Parallel Processing* (p. 1585)).

If you select the **Show More Options** button, FLUENT Launcher expands to reveal more options. Note that once FLUENT Launcher expands, the **Show More Options** button becomes the **Show Fewer Options** button, allowing you to hide the additional options.

Important

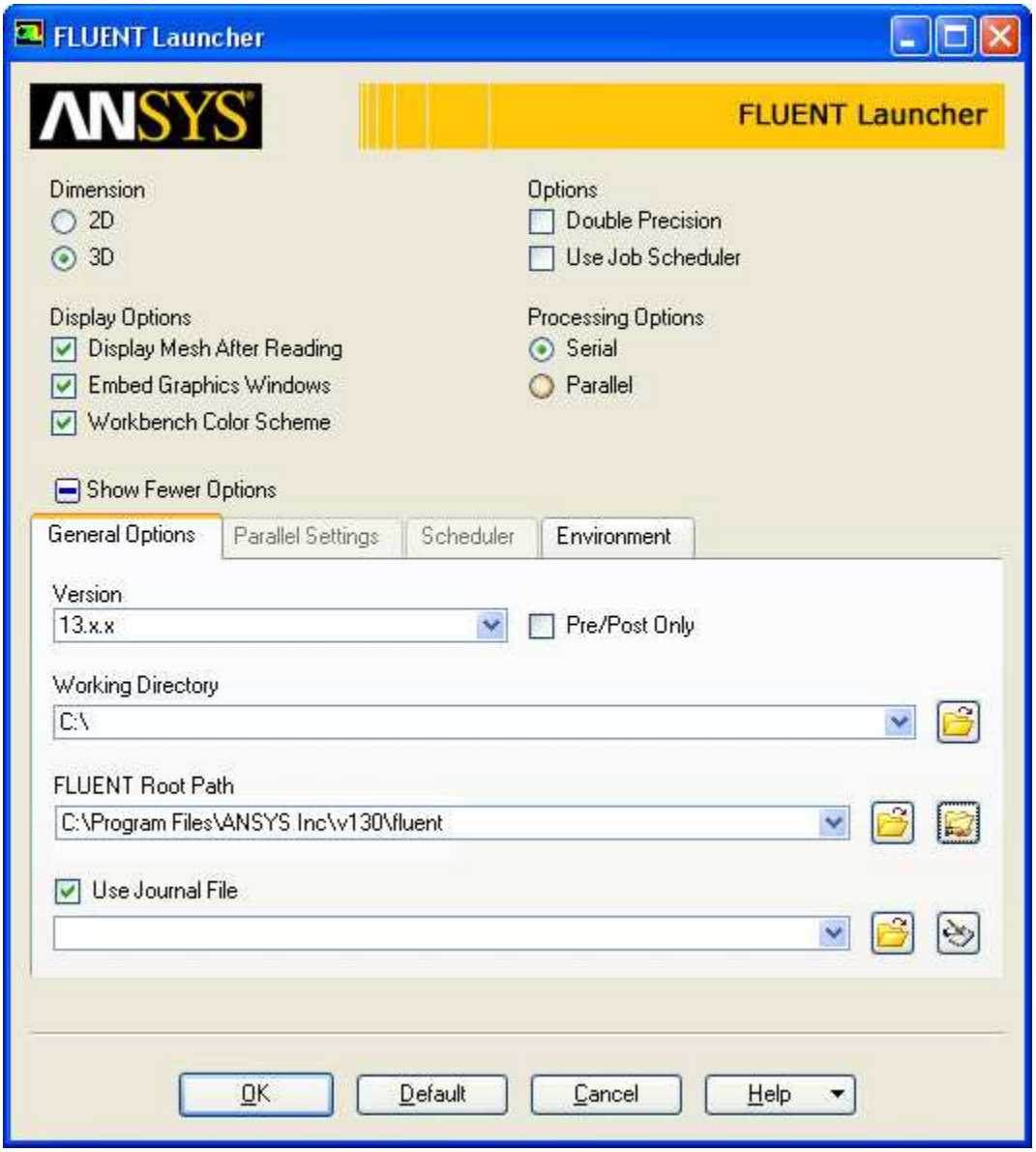
FLUENT Launcher also appears when you start ANSYS FLUENT within ANSYS Workbench. For more information, see the separate ANSYS FLUENT in Workbench User's Guide.

1.1.2.1. Setting General Options in FLUENT Launcher

The **General Options** tab allows you to specify generic settings for running ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.2 The General Options Tab of FLUENT Launcher



- Specify the version of ANSYS FLUENT by selecting the appropriate option in the **Version** drop-down list. The drop-down list contains all of the available versions of ANSYS FLUENT that exist in your ANSYS FLUENT installation.

  In addition, you start ANSYS FLUENT in full simulation mode. You can choose to run ANSYS FLUENT where only the set up or postprocessing capabilities are available by selecting the **Pre/Post Only** check box. The full ANSYS FLUENT simulation allows you to set up, solve and postprocess a problem, while **Pre/Post Only** allows you to set up or postprocess a problem, but will not allow you to perform calculations.

- Specify the path of your current working folder using the **Working Directory** field or click to browse through your directory structure. Note that a UNC path cannot be set as a working folder, and you need to map a drive to the UNC path (Windows only). The button automatically converts a local path to a UNC path if any matching shared directory is found (Windows only).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

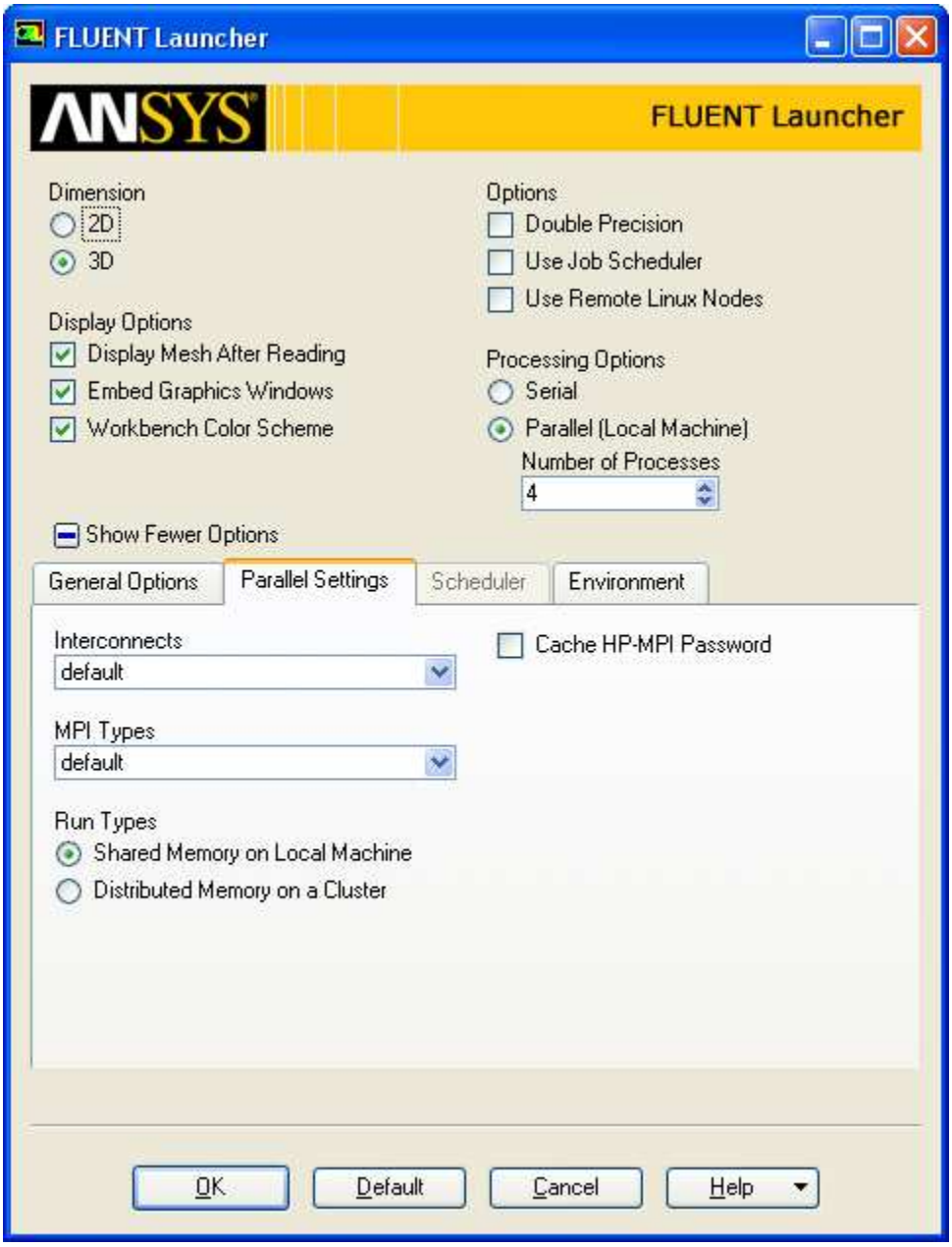
- Specify the location of where ANSYS FLUENT is installed on your system using the **FLUENT Root Path** field, or click to browse through your directory structure to locate the installation folder (trying to use the UNC path if applicable – Windows only). Once set, various fields in FLUENT Launcher (e.g., parallel settings, etc.) are automatically populated with the available options, depending on the ANSYS FLUENT installations that are available.
- Specify the path and name of a journal file by selecting the **Use Journal File** check box and entering the journal file location and name, or click to browse through your directory structure to locate the file. Using the journal file, you can automatically load the case, compile any user-defined functions, iterate until the solution converges, and write results to an output file.

1.1.2.2. Setting Parallel Options in FLUENT Launcher

The **Parallel Settings** tab allows you to specify settings for running ANSYS FLUENT in parallel. This tab is only available if you have selected **Parallel** under **Processing Options**. Once you select **Parallel**, you can specify the number of processes using the **Number of Processes** field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.3 The Parallel Settings Tab of FLUENT Launcher



For additional information about this tab, see *Starting Parallel ANSYS FLUENT Using FLUENT Launcher* (p. 1587).

1.1.2.3. Setting Remote Options in FLUENT Launcher

The **Remote** tab allows you to specify settings for running ANSYS FLUENT parallel simulations on Linux clusters, via the Windows interface.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.4 The Remote Tab of FLUENT Launcher

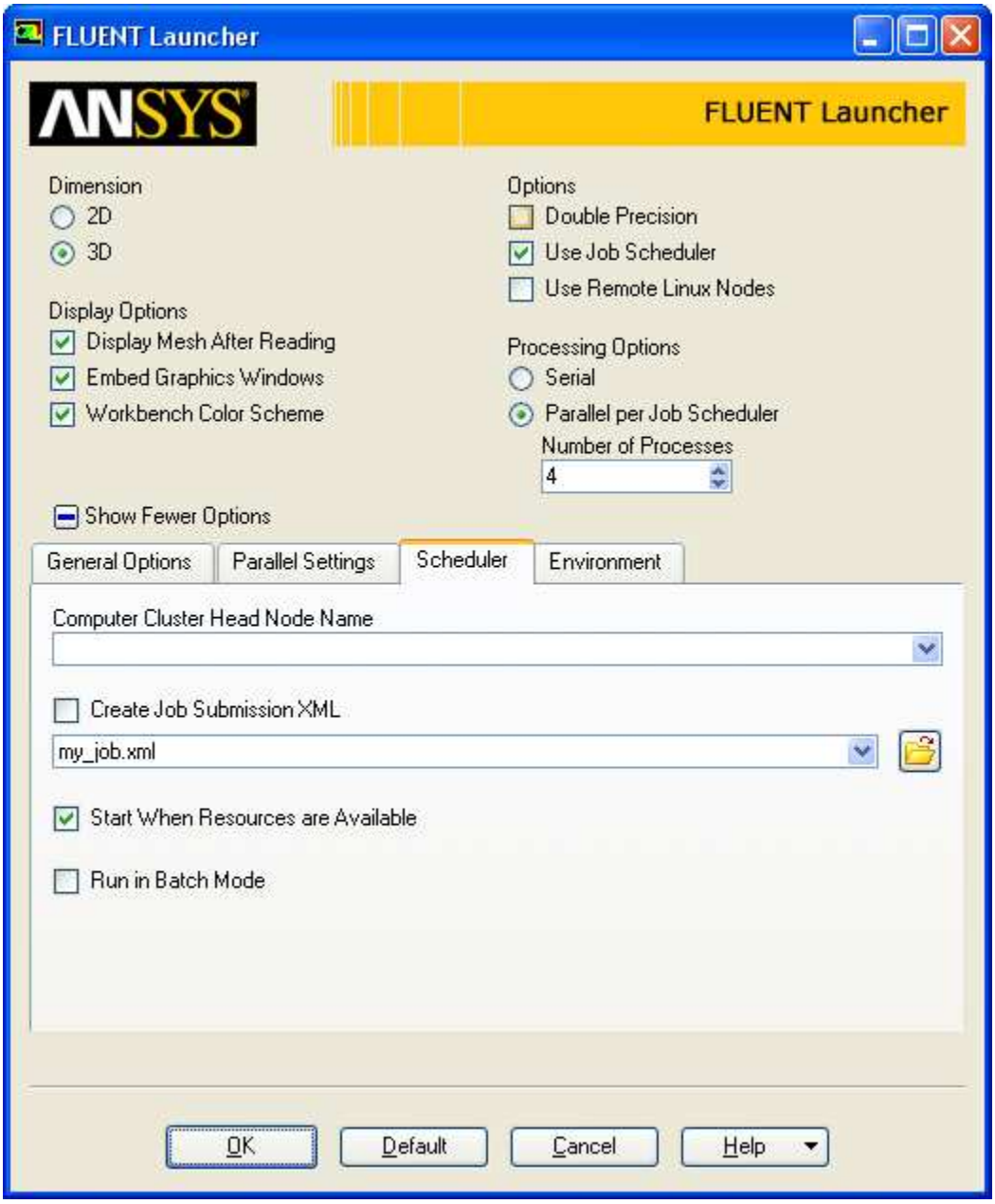
For additional information about this tab, see *Setting Additional Options When Running on Remote Linux Machines* (p. 1591).

1.1.2.4. Setting Scheduler Options in FLUENT Launcher

The **Scheduler** tab allows you to specify settings for running ANSYS FLUENT with various job schedulers (e.g., the Microsoft Job Scheduler for Windows, or LSF, SGE, and PBS Pro on Linux/UNIX). This tab is available if you have selected **Use Job Scheduler** under **Options**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.5 The Scheduler Tab of FLUENT Launcher (Windows Version)



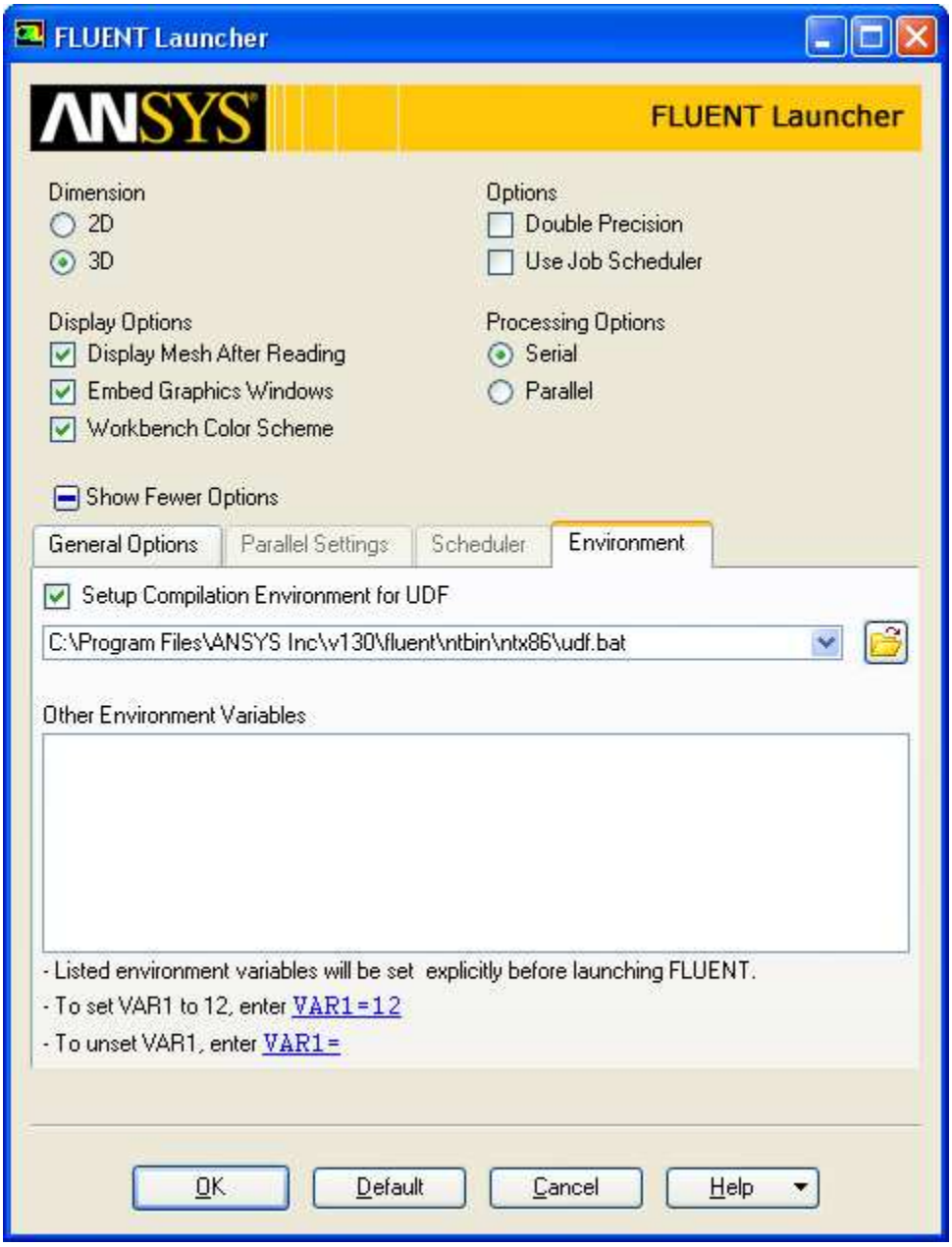
For additional information about this tab, see *Setting Parallel Scheduler Options in FLUENT Launcher* (p. 1589).

1.1.2.5. Setting Environment Options in FLUENT Launcher

The **Environment** tab allows you to specify compiler settings for compiling user-defined functions (UDFs) with ANSYS FLUENT (Windows only). The **Environment** tab also allows you to specify environment variable settings for running ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.6 The Environment Tab of FLUENT Launcher



Specify a batch file that contains UDF compilation environment settings by selecting the **Setup Compilation Environment for UDF** check box (enabled by default). Once selected, you can then enter a batch file name in the text field. By default, FLUENT Launcher uses the `udf.bat` file that is located in the folder where ANSYS FLUENT is installed. It is recommended that you keep the default batch file, which is tested with the latest MS Visual Studio C++ compilers at the time of the ANSYS FLUENT release date. For more information about compiling UDFs, see the separate ANSYS FLUENT UDF Manual.

Under **Other Environment Variables**, enter or edit license file or environment variable information in the text field. Using the **Default** button resets the default value(s).

1.1.3. Starting ANSYS FLUENT on a Windows System

There are two ways to start ANSYS FLUENT on a Windows system:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the Start button, select the All Programs menu, select the `ANSYS 13.0` menu, select the `Fluid Dynamics` menu, the select the `FLUENT` program item. (Note that if the default "ANSYS 13.0" program group name was changed when ANSYS FLUENT was installed, you will find the `FLUENT` menu item in the program group with the new name that was assigned, rather than in the `ANSYS 13.0` program group.) This option starts FLUENT Launcher (see *Starting ANSYS FLUENT Using FLUENT Launcher* (p. 2)).
- Start from a Command Prompt window by typing `fluent 2d` (for the 2D single-precision solver), `fluent 3d` (for the 3D single-precision solver), `fluent 2ddp` (for the 2D double-precision solver), or `fluent 3ddp` (for the 3D double-precision solver) at the prompt. Before doing so, however, you must first modify your user environment so that the Command utility will find `fluent`. You can do this by executing the `setenv.exe` program located in the ANSYS FLUENT home directory (e.g., `C:\Program Files\ANSYS Inc\v130\fluent\ntbin\ntx86`). This program will add the ANSYS FLUENT folder to your command search path.

  From the Command Prompt window, you can also start the parallel version of ANSYS FLUENT. To start the parallel version on `x` processors, type `fluent version -tx` at the prompt, replacing `version` with the desired solver version (`2d`, `3d`, `2ddp`, or `3ddp`) and `x` with the number of processors (e.g., `fluent 3d -t4` to run the 3D version on 4 processors). For information about the parallel version of ANSYS FLUENT for Windows, see *Starting Parallel ANSYS FLUENT on a Windows System* (p. 1594).

1.1.4. Starting ANSYS FLUENT on a Linux/UNIX System

There are two ways to start ANSYS FLUENT on a Linux/UNIX system:

- Start the solver from the command line without specifying a version, and then use FLUENT Launcher to choose the appropriate version along with other options. See *Starting ANSYS FLUENT Using FLUENT Launcher* (p. 2) for details.
- Start the appropriate version from the command line by typing `fluent2d` (for the 2D single-precision solver), `fluent3d` (for the 3D single-precision solver), `fluent 2ddp` (for the 2D double-precision solver), or `fluent 3ddp` (for the 3D double-precision solver) at the prompt.

You can also start the parallel version of ANSYS FLUENT from the command line. To start the parallel version on `x` processors, type `fluent version -tx` at the prompt, replacing `version` with the desired solver version (`2d`, `3d`, `2ddp`, or `3ddp`) and `x` with the number of processors (e.g., `fluent 3d -t4` to run the 3D version on 4 processors). See *Starting Parallel ANSYS FLUENT on a Linux/UNIX System* (p. 1599) for more information about starting the parallel solvers.

1.1.5. Command Line Startup Options

To obtain information about available startup options, you can type `fluent -help` before starting up the solver. *Table 1.1: Available Command Line Options for Linux/UNIX and Windows Platforms* (p. 10) lists the available command line arguments for Linux/UNIX and Windows. More detailed descriptions of these options can be found in the proceeding sections.

Table 1.1 Available Command Line Options for Linux/UNIX and Windows Platforms

Option	Platform	Description
`-cc`	all	Use the classic color scheme
`-ccp x`	Windows only	Use the Microsoft Job Scheduler where `x` is the head node name.
`-cnf=x`	all	Specify the hosts or machine list file

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Platform	Description
`-driver`	all	Sets the graphics driver (available drivers vary by platform – `opengl` or `x11` or `null` (Linux/UNIX) – `opengl` or `msw` or `null` (Windows))
`-env`	all	Show environment variables
`-fgw`	all	Disables the embedded graphics
`-g`	all	Run without the GUI or graphics (Linux/UNIX); Run with the GUI minimized (Windows)
`-gr`	all	Run without graphics
`-gu`	all	Run without the GUI but with graphics (Linux/UNIX); Run with the GUI minimized but with graphics (Windows)
`-help`	all	Display command line options
`-hidden`	Windows only	Run in batch mode
`-host_ip=host:ip`	all	Specify the IP interface to be used by the host
`-i journal`	all	Reads the specified journal file
`-lsf`	Linux/UNIX only	Run ANSYS FLUENT using LSF
`-mpi=`	all	Specify MPI implementation
`-mpitest`	all	Will launch an MPI program to collect network performance data
`-nm`	all	Do not display mesh after reading
`-pcheck`	Linux/UNIX only	Checks all nodes
`-post`	all	Run the ANSYS FLUENT post-processing-only executable
`-p<ic>`	all	Choose the interconnect `<ic>`= `default` or `myr` or `inf`
`-r`	all	List all releases installed
`-rx`	all	Specify release number
`-sge`	Linux/UNIX only	Run ANSYS FLUENT under Sun Grid Engine
`-sge queue`	Linux/UNIX only	Name of the queue for a given computing grid
`-sgeckpt ckpt_obj`	Linux/UNIX only	Set checkpointing object to `ckpt_obj` for SGE
`-sgepe fluent_pe min_n-max_n`	Linux/UNIX only	Set the parallel environment for SGE to `fluent_pe`, `min_n` and `max_n` are number of min and max nodes requested
`-tx`	all	Specify the number of processors `x`

1.1.5.1. Graphics Options

`fluent -driver` allows you to specify the graphics driver to be used in the solver session. For example, on Linux/UNIX you can specify `fluent -driver opengl`, `fluent -driver x11`, and `fluent -driver null`. These options are described in detail in *Hiding the Graphics Window Display* (p. 1410). On Windows you can specify `fluent -driver opengl` and `fluent -driver msw` to enable graphics display. Using `msw` instead of `opengl` instructs ANSYS FLUENT to use the Operating Systems Windows driver rather than the hardware OpenGL driver.

`fluent -cc` will run Cortex using the classic black background color in the graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`fluent -fgw` will run Cortex without the graphics window embedded in the application (e.g., "floating").

`fluent -g` will run Cortex without graphics and without the graphical user interface. This option is useful if you are not on an X Window display or if you want to submit a batch job.

`fluent -gr` will run Cortex without graphics. This option can be used in conjunction with the `-i journal` option to run a job in "background" mode.

`fluent -gu` will run Cortex without the graphical user interface but will display the graphics window(s). (On Windows systems, `fluent -gu` will run ANSYS FLUENT, keeping it in a minimized window; if you maximize the window, the GUI *will* be available.)

To start the solver and immediately read a journal file, type `fluent -i journal`, replacing `journal` with the name of the journal file you want to read.

`fluent -nm` will run Cortex without displaying the mesh in the graphics window.

1.1.5.2. Parallel Options

These options are used in association with the parallel solver.

`-ccp x` (where `x` is the name of the head node) runs the parallel job through the Microsoft Job Scheduler as described in *Starting Parallel ANSYS FLUENT with the Microsoft Job Scheduler* (p. 1597).

`-cnf=x` (where `x` is the name of a hosts file) spawns a compute node on each machine listed in the hosts file. Otherwise, you can spawn the processes as described in *Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options* (p. 1595) or *Starting Parallel ANSYS FLUENT on a Linux/UNIX System Using Command Line Options* (p. 1599).

`-host_ip=host:ip` specifies the IP interface to be used by the host process (Linux/UNIX only).

`-mpi=mpi` specifies the MPI to be used. You can skip this flag if you choose to use the default MPI.

`-mpitest` runs the mpitest program instead of ANSYS FLUENT to test the network.

`-p<ic>` specifies the use of parallel interconnect `<ic>`, where `<ic>` can be any of the interconnect listed in *Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options* (p. 1595) or *Starting Parallel ANSYS FLUENT on a Linux/UNIX System Using Command Line Options* (p. 1599).

`-pcheck` checks the network connections before spawning compute nodes (Linux/UNIX only).

`-tx` specifies that `x` processors are to be used. For more information about starting the parallel version of ANSYS FLUENT, see *Starting Parallel ANSYS FLUENT on a Windows System* (p. 1594) or *Starting Parallel ANSYS FLUENT on a Linux/UNIX System* (p. 1599).

1.1.5.3. Postprocessing Option

`fluent -post` will run a version of the solver that allows you to set up a problem or perform postprocessing, but will not allow you to perform calculations. Running ANSYS FLUENT for pre- and postprocessing requires you to use the -post flag on startup. To use this option on Linux/UNIX, launch ANSYS FLUENT by adding the `-post` flag after the version number, for example,

```
fluent 3d -post
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use this same feature from the graphical interface on Windows or Linux/UNIX, select the **Pre/Post** option in the **General** tab of FLUENT Launcher, as described in *Starting ANSYS FLUENT Using FLUENT Launcher* (p. 2).

1.1.5.4. SGE Options

The `-sge` option runs ANSYS FLUENT under Sun Grid Engine (SGE) software, and allows you to use the features of this software to manage your distributed computing resources. Other options that can be employed in conjunction with `-sge` are `-sge, requested`, `-sgeq queue`, `-sgeckpt ckpt_obj`, and `-sgepe fluent_pe min_n-max_n`.

For a detailed explanation of these options, go to the Documentation page (`http://fluentusers.com/fluent/doc/doc_f.htm`) on the User Services Center (`www.fluentusers.com`) and locate the Load Management Documentation page.

1.1.5.5. LSF Options

The `-lsf` option allows you to run ANSYS FLUENT under Platform Computing Corporation's LSF software, and thereby take advantage of the checkpointing features of that load management tool. For further details about using the `-lsf` option, go to the Documentation page (`http://fluentusers.com/fluent/doc/doc_f.htm`) on the User Services Center (`www.fluentusers.com`) and locate the Load Management Documentation page.

1.1.5.6. Version and Release Options

Typing `fluent version -r`, replacing `version` with the desired version, will list all releases of the specified version.

`fluent -rx` will run release `x` of ANSYS FLUENT. You may specify a version as well, or you can wait and specify the version when prompted by the solver.

`fluent -env` will list all environment variables before running ANSYS FLUENT.

1.1.5.7. Other Startup Options

There are other startup options that are not listed when you type the `fluent -help` command. These options can be used to customize your graphical user interface. For example, to change the ANSYS FLUENT window size and position you can either modify the.Xdefaults file (Linux/UNIX only) described in *Customizing the Graphical User Interface (UNIX Systems Only)* (p. 40), or you can simply type the following command at startup:

```
fluent [version] [-geometry] [XXxYY+00-50]
```

where XX and YY are the width and height in pixels, respectively, and +00-50 is the position of the window.

Therefore, typing `fluent 3d -geometry 700x500+20-400` will start the 3D version of ANSYS FLUENT, sizing the ANSYS FLUENT console to 700x500 pixels and positioning it on your monitor screen at +20-400.

1.1.6. Remote Simulation Facility (RSF)

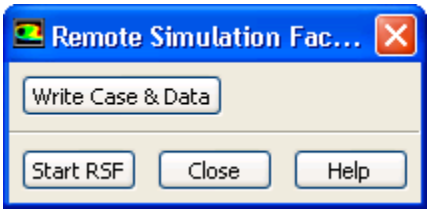
ANSYS FLUENT's Remote Simulation Facility (RSF) is a highly secure mechanism for executing large, complex simulations at a remote data center. You can use ANSYS FLUENT to access the RSF using the **File** menu.

File → **RSF...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select this option, ANSYS FLUENT displays the **Remote Simulation Facility** dialog box (*Figure 1.7* (p. 14)).

Figure 1.7 The Remote Simulation Facility Dialog Box



If a case and data file are already loaded into ANSYS FLUENT, the **Write Case & Data** button will be active, which will allow you to write out the case and data files for the current session. You must save your case and data files in order to use them on the RSF. Once you save the case and data files and launch the RSF, you can exit from ANSYS FLUENT.

To start an RSF session, click **Start RSF** to open your web browser and load the page for the ANSYS FLUENT Remote Simulation Facility (RSF).

For more information about the RSF, see `www.ansys.com/support/remotesimulation`.

1.2. Running ANSYS FLUENT in Batch Mode

ANSYS FLUENT can be used interactively, with input from and display to your computer screen, or it can be used in a batch or background mode in which inputs are obtained from and outputs are stored in files. Generally you will perform problem setup, initial calculations, and postprocessing of results in an interactive mode. However, when you are ready to perform a large number of iterative calculations, you may want to run ANSYS FLUENT in batch or background mode. This allows the computer resources to be prioritized, enables you to control the process from a file (eliminating the need for you to be present during the calculation), and also provides a record of the calculation history (residuals) in an output file. While the procedures for running ANSYS FLUENT in a batch mode differ depending on your computer operating system, *Background Execution on Linux/UNIX Systems* (p. 14) provides guidance for running in batch/background on Linux/UNIX systems, and *Background Execution on Windows Systems* (p. 15) provides guidance for running in batch/background on Windows systems.

For additional information, please see the following sections:

1.2.1. Background Execution on Linux/UNIX Systems
1.2.2. Background Execution on Windows Systems
1.2.3. Batch Execution Options

1.2.1. Background Execution on Linux/UNIX Systems

To run ANSYS FLUENT in the background in a C-shell (csh) on a Linux/UNIX system, type a command of the following form at the system-level prompt:

```
fluent 2d -g < inputfile > & outputfile &
```

or in a Bourne/Korn-shell, type:

```
fluent 2d -g < inputfile > outputfile 2>&1 &
```

In these examples,

- `fluent` is the command you type to execute ANSYS FLUENT interactively.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `-g` indicates that the program is to be run without the GUI or graphics (see *Starting ANSYS FLUENT* (p. 1)).
- `inputfile` is a file of ANSYS FLUENT commands that are identical to those that you would type interactively.
- `outputfile` is a file that the background job will create and which will contain the output that ANSYS FLUENT would normally print to the screen (e.g., the menu prompts and residual reports).
- & tells the Linux/UNIX system to perform this task in background and to send all standard system errors (if any) to `outputfile`.

The file `inputfile` can be a journal file created in an earlier ANSYS FLUENT session, or it can be a file that you have created using a text editor. In either case, the file must consist only of text interface commands (since the GUI is disabled during batch execution). A typical `inputfile` is shown below:

```
; Read case file
rc example.cas
; Initialize the solution
/solve/initialize/initialize-flow
; Calculate 50 iterations
it 50
; Write data file
wd example50.dat
; Calculate another 50 iterations
it 50
; Write another data file
wd example100.dat
; Exit ANSYS FLUENT
exit
yes
```

This example file reads a case file `example.cas`, initializes the solution, and performs 100 iterations in two groups of 50, saving a new data file after each 50 iterations. The final line of the file terminates the session. Note that the example input file makes use of the standard aliases for reading and writing case and data files and for iterating. (`it` is the alias for `/solve/iterate`, `rc` is the alias for `/file/read-case`, `wd` is the alias for `/file/write-data`, etc.) These predefined aliases allow you to execute commonly-used commands without entering the text menu in which they are found. In general, ANSYS FLUENT assumes that input beginning with a `/` starts in the top-level text menu, so if you use any text commands for which aliases do not exist, you must be sure to type in the complete name of the command (e.g., `/solve/initialize/initialize-flow`). Note also that you can include comments in the file. As in the example above, comment lines must begin with a `;` (semicolon).

An alternate strategy for submitting your batch run, as follows, has the advantage that the `outputfile` will contain a record of the commands in the `inputfile`. In this approach, you would submit the batch job in a C-shell using:

```
fluent 2d -g -i inputfile  >& outputfile &
```

or in a Bourne/Korn-shell using:

```
fluent 2d -g -i inputfile > outputfile 2>&1 &
```

1.2.2. Background Execution on Windows Systems

To run ANSYS FLUENT in the background on a Windows system, the following commands can be used:

```
fluent 3d -g -i journal

fluent 3d -g -wait -i journal

fluent 3d -g -hidden -i journal
```

In these examples,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `fluent` is the command you type to execute ANSYS FLUENT interactively.
- `-g` indicates that the program is to be run minimized in the task bar.
- `-i journal` reads the specified journal file.
- `-wait` is the command you type in a DOS batch file or some other script in a situation where the script needs to wait until ANSYS FLUENT has completed its run.
- `-hidden` is similar to the `-wait` command, but also executes ANSYS FLUENT completely hidden and noninteractively.

To get an output (or transcript) file while running ANSYS FLUENT in the background on a Windows system, the journal file must contain the following command to write a transcript file:

```
; start transcript file
 /file/start-transcript outputfile.trn
```

where the `outputfile` is a file that the background job will create and which will contain the output that ANSYS FLUENT would normally print to the screen (e.g., the menu prompts and residual reports).

See *Creating and Reading Journal Files* (p. 73) for details about journal files. See *Creating Transcript Files* (p. 75) for details about transcript files.

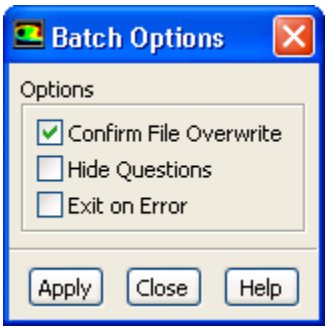
1.2.3. Batch Execution Options

During a typical session, ANSYS FLUENT may require feedback from you in the event of a problem it encounters. ANSYS FLUENT usually communicates problems or questions through the use of **Error** dialog boxes, **Warning** dialog boxes, or **Question** dialog boxes. While executing ANSYS FLUENT in batch mode, you may want to suppress this type of interaction in order to, for example, create journal files more easily.

There are three common batch configuration options available to you when running ANSYS FLUENT in batch mode. You can access these options using the **Batch Options** dialog box (see *Batch Options Dialog Box* (p. 2073)).

File → **Batch Options...**

Figure 1.8 The Batch Options Dialog Box



The **Batch Options** dialog box contains the following items:

Confirm File Overwrite
determines whether ANSYS FLUENT confirms a file overwrite. This option is turned on by default.

Hide Questions
allows you to hide **Question** dialog boxes. This option is turned off by default.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exit on Error

allows you to automatically exit from batch mode when an error occurs. This option is turned off by default.

Note that these options are also available in the `file/set-batch-options` command in the text interface.

`file` → `set-batch-options`

Any combination of these options can be turned on or off at any given time prior to running in batch mode.

Important

Batch option settings are *not* saved with case files. They are meant to apply for the duration of the current ANSYS FLUENT session only. If you read in additional mesh or case files during this session, the batch option settings will not be altered. As batch options are not saved with case files, journal files developed for use in batch mode should begin by enabling the desired batch option settings (if different from the default settings).

1.3. Checkpointing an ANSYS FLUENT Simulation

The checkpointing feature of ANSYS FLUENT allows you to save case and data files while your simulation is running. While similar to the autosave feature of ANSYS FLUENT (*Automatic Saving of Case and Data Files* (p. 65)), which allows you to save files throughout a simulation, checkpointing allows you slightly more control in that you can save an ANSYS FLUENT job even after you have started the job and did not set the autosave option. Checkpointing also allows you to save case and data files and then exit out of ANSYS FLUENT. This feature is especially useful when you need to stop an ANSYS FLUENT job abruptly and save its data.

There are two different ways to checkpoint an ANSYS FLUENT simulation, depending upon how the simulation has been started.

1. ANSYS FLUENT running under LSF or SGE

   ANSYS FLUENT is integrated with load management tools like LSF and SGE. These two tools allow you to checkpoint any job running under them. You can use the standard method provided by these tools to checkpoint the ANSYS FLUENT job.

   For more information on using ANSYS FLUENT and SGE or LSF, go to the Documentation page (`http://fluentusers.com/fluent/doc/doc_f.htm`) on the User Services Center (`www.fluentusers.com`) and locate the Load Management Documentation page.

2. Independently running ANSYS FLUENT

   When not using tools such as LSF or SGE, a different checkpointing mechanism can be used when running an ANSYS FLUENT simulation. You can checkpoint an ANSYS FLUENT simulation while iterating/time-stepping, so that ANSYS FLUENT saves the case and data files and then continues the calculation, or so that ANSYS FLUENT saves the case and data files and then exits.

   - Saving case and data files and continuing the calculation:

     On Linux/UNIX, create a file called `check-fluent`, i.e.,

     ```
     /tmp/check-fluent
     ```

     On Windows, create a file called `check-fluent.txt`, i.e.,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
C:%USERPROFILE%\check-fluent.txt
```

- Saving case and data files and exiting ANSYS FLUENT:

  On Linux/UNIX, create a file called `exit-fluent`, i.e.,

  ```
  /tmp/exit-fluent
  ```

  On Windows, create a file called `exit-fluent.txt`, i.e.,

  ```
  %USERPROFILE%\exit-fluent.txt
  ```

The saved case and data files will have the current iteration number appended to their file names.

ANSYS FLUENT offers an alternate way to checkpoint an unsteady simulation. While the default behavior is to checkpoint the simulation at the end of the current iteration, for unsteady simulations you have the option of completing all of the iterations in the current time-step before checkpointing. This can be set by entering the following Scheme command prior to running the unsteady simulation:

```
(ckpt/time-step? #t)
```

Now when you save the checkpoint file (as described previously), the case and data file will be saved at the end of the current time-step and named accordingly. To switch back to the default checkpointing mechanism at the end of the current iteration, use the following Scheme command:

```
(ckpt/time-step? #f)
```

Important

Note that the `(ckpt/time-step? #t)` command will have the effect only in the case of an unsteady simulation.

To change the default location of the saved case and data files, you can use the following Scheme commands:

```
(set! checkpoint/check-filename pathname)
```

and

```
(set! checkpoint/exit-filename pathname)
```

where `pathname` is the path you wish to set as the new default location of the saved case and data files.

1.4. Cleaning Up Processes From an ANSYS FLUENT Simulation

ANSYS FLUENT lets you easily remove extraneous processes in the event that an ANSYS FLUENT simulation needs to be stopped.

When a session is started, ANSYS FLUENT creates a `cleanup-fluent` script file. The script can be used to clean up all ANSYS FLUENT-related processes. ANSYS FLUENT creates the cleanup-script file in the current working folder with a file name that includes the machine name and the process identification number (PID) (e.g., `cleanup-fluent-mymachine-1234`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the current directory does not possess the proper write permissions, then ANSYS FLUENT will write the cleanup-script file to your home directory.

If, for example, ANSYS FLUENT is started on a machine called `thor` and the process identification number is 32895, ANSYS FLUENT will create a cleanup-script called `cleanup-fluent-thor-32895` in the current folder. To run the cleanup-script, and clean up all ANSYS FLUENT processes related to your session, on Linux or UNIX platforms, type the following command in the console window:

```
sh cleanup-fluent-thor-32895
```

Or, if the shell script already has executable permissions, simply type:

```
cleanup-fluent-thor-32895
```

To clean up extraneous ANSYS FLUENT processes on Windows (serial or parallel), double-click the corresponding batch file (e.g., `cleanup-fluent-thor-32895.bat`) that ANSYS FLUENT generates at the beginning of each session.

Important

During a normal run, this file will be deleted automatically after exiting ANSYS FLUENT. In abnormal situations, you may use this batch file to clean up the ANSYS FLUENT processes. Once an ANSYS FLUENT session has been closed, you can safely delete any left over cleanup scripts from your working folder.

1.5. Exiting the Program

You can exit ANSYS FLUENT by selecting **Exit** in the **File** pull-down menu. If the present state of the program has not been written to a file, a **Question** dialog box will open to confirm if you want to proceed. You can cancel the exit and write the appropriate file(s) or you can continue to exit without saving the case or data.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Graphical User Interface (GUI)

The ANSYS FLUENT graphical interface consists of a menu bar to access the menus, a toolbar, a navigation pane, a task page, a graphics toolbar, graphics windows, and a console, which is a textual command line interface (described in *Text User Interface (TUI)* (p. 47)). You will have access to the dialog boxes via the task page or the menus.

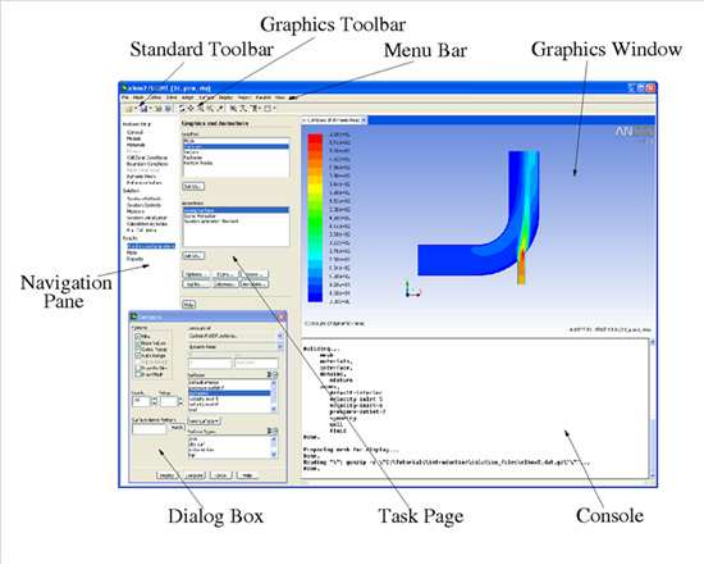
2.1. GUI Components
2.2. Customizing the Graphical User Interface (UNIX Systems Only)
2.3. Using the GUI Help System

2.1. GUI Components

The graphical user interface (GUI) is made up of seven main components: the menu bar, toolbars, a navigation pane, task pages, a console, dialog boxes, and graphics windows. When you use the GUI, you will be interacting with one of these components at all times. *Figure 2.1* (p. 21) is a sample screen shot showing all of the GUI components. The seven GUI components are described in detail in the subsequent sections.

On Linux/UNIX systems, the attributes of the GUI (including colors and text fonts) can be customized to better match your platform environment. This is described in *Customizing the Graphical User Interface (UNIX Systems Only)* (p. 40).

Figure 2.1 The GUI Components



For additional information, please see the following sections:

2.1.1. The Menu Bar
2.1.2. Toolbars
2.1.3. The Navigation Pane
2.1.4. Task Pages
2.1.5. The Console
2.1.6. Dialog Boxes

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.7. Graphics Windows

2.1.1. The Menu Bar

The menu bar organizes the GUI menu hierarchy using a set of pull-down menus. A pull-down menu contains items that perform commonly executed actions. *Figure 2.2* (p. 22) shows the ANSYS FLUENT menu bar. Menu items are arranged to correspond to the typical sequence of actions that you perform in ANSYS FLUENT (i.e., from left to right and from top to bottom).

Figure 2.2 The ANSYS FLUENT Menu Bar

To select a pull-down menu item with the mouse, follow the procedure outlined below:

1. Move the pointer to the name of the pull-down menu.
2. Click the left mouse button to display the pull-down menu.
3. Move the pointer to the item you wish to select and click it.

In addition to using the mouse, you can also select a pull-down menu item using the keyboard. If you press the `<Alt>` key, each pull-down menu label or menu item will display one underlined character, known as the mnemonic. If you then press the mnemonic character of a pull-down menu, the associated menu will be displayed (note that the mnemonic character is not case sensitive). After the pull-down menu is selected and displayed, you can type a mnemonic character associated with an item to select that item. For example, to display the **Help** menu and select the **Using Help...** option, press `<Alt>`, then `h`, and then `h` again. If at any time you wish to cancel a menu selection while a pull-down menu is displayed, you can press the `<Esc>` key.

2.1.2. Toolbars

The ANSYS FLUENT GUI includes toolbars located within the application window. These toolbars provide shortcuts to performing common tasks in ANSYS FLUENT. By default, the toolbars are docked to the ANSYS FLUENT interface but can also be detached and moved to a new location. You can detach a toolbar by clicking the left mouse button on the outer portion of it, holding down the mouse, and dragging the toolbar to a new location. To move the detached toolbar, select the title bar and drag the toolbar to a new position in the application window. Once detached, the toolbars can be restored to their location in the interface by double-clicking the title region of the toolbar.

Important

Toolbars that are detached or moved to a new location will return to their original positions each time ANSYS FLUENT is launched.

The small arrow button in some ANSYS FLUENT toolbars can be used to access additional functionality in ANSYS FLUENT. For instance, there are additional selections available when you click the small arrow in the standard toolbar.

The ANSYS FLUENT graphical user interface includes a standard toolbar and a graphics toolbar.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.2.1. The Standard Toolbar

The standard toolbar (*Figure 2.3* (p. 23)) contains options for working with ANSYS FLUENT case files, saving images, and accessing the ANSYS FLUENT documentation.

Figure 2.3 The Standard Toolbar

The following is a brief description of each of the standard toolbar options.

- **Read a file** allows you to read in a mesh, open existing ANSYS FLUENT case files, and other file types using a file selection dialog box. Here, you can browse through your collection of folders, and locate a file. For more information, see *Reading and Writing Case and Data Files* (p. 63).

- **Write a file** saves the current ANSYS FLUENT case, data, or other file types. For more information, see *Reading and Writing Case and Data Files* (p. 63).

- **Save Picture** allows you to capture an image of the active graphics window. For more information, see *Saving Picture Files* (p. 117).

- **Help** allows you to access the ANSYS FLUENT User's Guide for help topics. For more information, see *Using the GUI Help System* (p. 41).

2.1.2.2. The Graphics Toolbar

The graphics toolbar (*Figure 2.4* (p. 23)) contains options that allow you to modify the way in which you view your model or select objects in the graphics window.

Figure 2.4 The Graphics Toolbar

The following is a description of each of the graphics toolbar options.

- **Rotate View** lets you rotate your model about a central point in the graphics window. For more information, see *Button Functions* (p. 1427).

- **Pan** allows you to pan horizontally or vertically across the view using the left mouse button. For more information, see *Button Functions* (p. 1427).

- **Zoom In/Out** allows you to zoom into and out of the model by holding the left mouse button down and moving the mouse down or up. For more information, see *Button Functions* (p. 1427). You can also roll the view by holding the left mouse button down and moving the mouse left or right.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Zoom to Area** allows you to focus on any part of your model. After selecting this option, position the mouse pointer at a corner of the area to be magnified, hold down the left mouse button and drag open a box to the desired size, and then release the mouse button. The enclosed area will then fill the graphics window. Note that you must drag the mouse to the right in order to zoom in. To zoom out, you must drag the mouse to the left For more information, see *Button Functions* (p. 1427).

- **Print information about selected item** allows you to select items from the graphics windows and request information about displayed scenes. This behaves as a mouse probe button. For more information, see *Button Functions* (p. 1427).

- **Fit to Window** adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.

- **Isometric view** contains a drop-down of views, allowing you to display the model from the direction of the vector equidistant to all three axes, as well as in different axes orientations.

- **Arrange the workspace** provides you with several application window layout options. For example, you can choose to hide certain windows, or view multiple graphics windows. This is essentially the shortcut to the **View** menu. For information about the various layouts, see *Viewing the Application Window* (p. 1428).

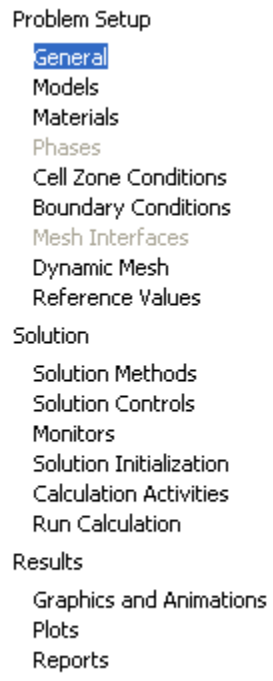
- **Arrange the graphics window layout** allows you to specify the number and layout of the graphics windows, when they are embedded in the ANSYS FLUENT application window. You can have up to four graphics window embedded at one time. This is essentially a shortcut to the **View/Graphics Window Layout** menu. See *Viewing the Application Window* (p. 1428) for further details.

2.1.3. The Navigation Pane

The navigation pane, located on the left side of the ANSYS FLUENT GUI, contains a list of task pages, as shown in *Figure 2.5* (p. 25).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.5 The ANSYS FLUENT Navigation Pane



The list consists of **Problem Setup** task pages, **Solution**-related activities, and a **Results** section for postprocessing.

When any of the items under **Problem Setup**, **Solution**, or **Results** is highlighted, the **task page** (*Task Pages* (p. 26)) will be displayed to the right of the navigation pane. The items in the navigation pane are listed in the order in which you would normally set up, solve, and postprocess a case.

Using the navigation pane is an alternative to using the menu bar. For example, there are two ways you can access the **Viscous Model** dialog box (see *Viscous Model Dialog Box* (p. 1645)). To access it using the navigation pane, highlight **Models** in the navigation pane by clicking on it with your left mouse button. The **Models** task page (see *Models Task Page* (p. 1639)) will appear to the right of the navigation pane. Select **Viscous** from the **Models** list and click the **Edit...** button (or simply double-click on **Viscous**) to open the **Viscous Model** dialog box (see *Viscous Model Dialog Box* (p. 1645)).

Models → **Viscous** → **Edit...**

You can also access the **Viscous Model** dialog box (see *Viscous Model Dialog Box* (p. 1645)) using the following menu path:

Define → **Models...**

This will open the **Models** task page (see *Models Task Page* (p. 1639)), where you will select **Viscous** from the **Models** list and click the **Edit...** button (or simply double-click on **Viscous**) to open the **Viscous Model** dialog box (see *Viscous Model Dialog Box* (p. 1645)).

Note that while most of the task pages can be accessed using the navigation pane, there are some dialog boxes that can only be accessed using the menu path. For example, to access the **Custom Field Function Calculator** dialog box (see *Custom Field Function Calculator Dialog Box* (p. 2106)), use the following menu path:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define → Custom Field Functions...

2.1.4. Task Pages

Task pages appear on the right side of the navigation pane when an item is highlighted in the navigation pane (see *Figure 2.1* (p. 21)). The expected workflow is that you travel down the navigation pane, setting the controls provided in each task page until you are ready to run the calculation. Note that you can access the task pages through the menu items, as described in the example in *The Navigation Pane* (p. 24).

Some of your setup will occur in dialog boxes, while others in task pages. For example, if **General** is selected in the navigation pane, this task page is displayed. Global settings are made in this task page, which are saved to the case definition.

Each task page has a **Help** button. Clicking this button opens the related help topic in the Reference Guide. See *Using the GUI Help System* (p. 41) for more information.

2.1.5. The Console

The console is located below the graphics window, as shown in *Figure 2.1* (p. 21). ANSYS FLUENT communicates with you through the console. It is used to display various kinds of information (i.e., messages relating to meshing or solution procedures, etc.). ANSYS FLUENT saves a certain amount of information that is written to the console into memory. You can review this information at any time by using the scroll bar on the right side of the console. The size of the console can be adjusted by raising or lowering the bottom frame of the graphics window.

The console is similar in behavior to "xterm" or other UNIX command shell tools, or to the MS-DOS Command Prompt window on Windows systems. It allows you to interact with the TUI menu. More information on the TUI can be found in *Text User Interface (TUI)* (p. 47).

The console accepts a "break" command (pressing the `<Ctrl>` key and the `c` key at the same time) to let you interrupt the program while it is working. It also lets you perform text copy and paste operations between the console and other X Window (or Windows) applications that support copy and paste. The following steps show you how to perform a copy and paste operation on a Windows system:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Press the `<Ctrl>` and `<Insert>` keys at the same time.
6. Move the pointer to the target window and click the left mouse button.
7. Press the `<Ctrl>` and `v` keys at the same time.

On a UNIX/Linux system, you will follow the steps below to copy text to the clipboard:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Move the pointer to the target window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Press the middle mouse button to “paste” the text.

2.1.6. Dialog Boxes

There are two types of dialog boxes in ANSYS FLUENT. Some dialog boxes are used to perform simple input/output tasks, such as issuing warning and error messages, or asking a question requiring a yes or no answer. Other forms of dialog boxes allow you to perform more complicated input tasks.

A dialog box is a separate “temporary” window that appears when ANSYS FLUENT needs to communicate with you, or when various types of input controls are employed to set up your case. The types of controls you will see are described further in this section.

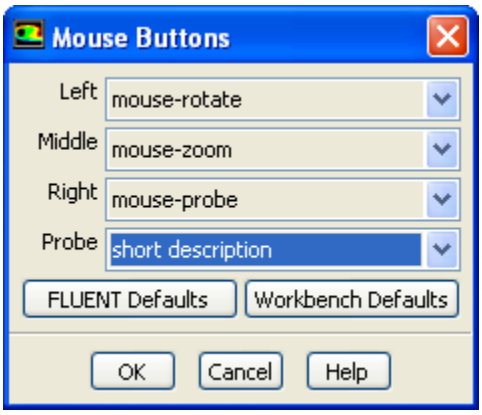
When you have finished entering data in a dialog box’s controls, you will need to apply the changes you have made, or cancel the changes, if desired. For this task, each dialog box falls into one of two behavioral categories, depending on how it was designed.

The first category of dialog boxes is used in situations where it is desirable to apply the changes and immediately close the dialog box. This type of dialog box includes two button controls as described below:

OK
applies any changes you have made to the dialog box, then closes the dialog box.

Cancel
closes the dialog box, ignoring any changes you have made.

An example of this type of dialog box is shown in the following figure:



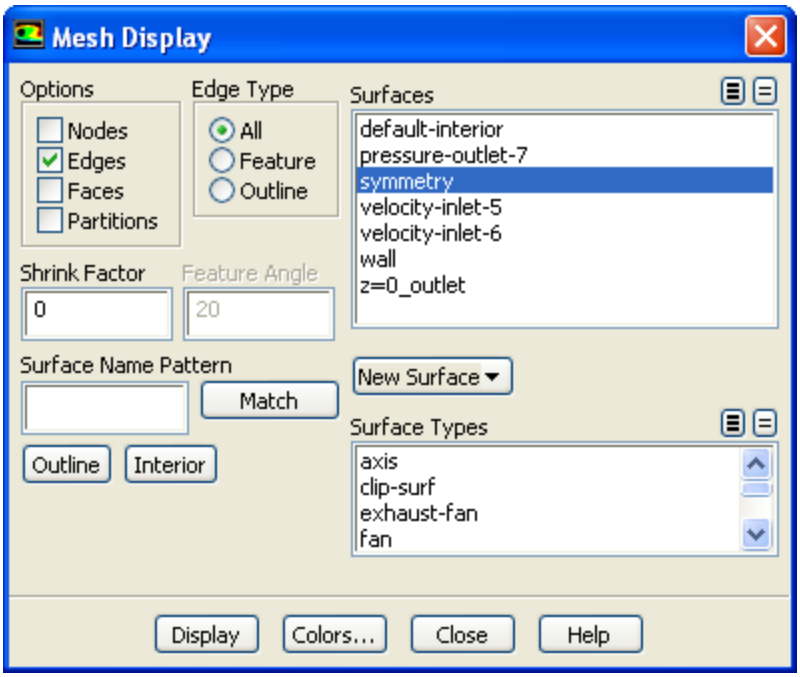
The other category of dialog boxes is used in situations where it is desirable to keep the dialog box displayed on the screen after changes have been applied. This makes it easy to quickly go back to that dialog box and make more changes. Dialog boxes used for postprocessing and mesh adaption often fall into this category. This type of dialog box includes two button controls as described below:

Apply
applies any changes you have made to the dialog box, but does not close the dialog box. The name of this button is often changed to something more descriptive. For example, many of the postprocessing dialog boxes use the name **Display** for this button, and the adaption dialog boxes use the name **Adapt**.

Close
closes the dialog box.

An example of this type of dialog box is shown in the following figure:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All dialog boxes include the following button used to access online help:

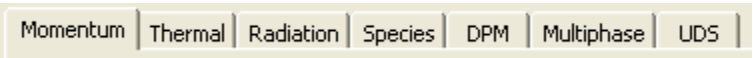
Help

displays information about the controls in the dialog box. The help information will appear in your web browser.

2.1.6.1. Input Controls

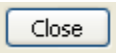
Each type of input control utilized by the dialog boxes is described below. Note that the examples shown here are for a Windows system; if you are working on a Linux/UNIX system, your dialog box controls may look slightly different, but they will work exactly as described here.

2.1.6.1.1. Tabs



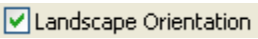
Much like the tabs on a notebook divider, tabs in dialog boxes are used to mark the different sections into which a dialog box is divided. A dialog box that contains many controls may be divided into different sections to reduce the amount of screen space it occupies. You can access each section of the dialog box by "clicking" the left mouse button on the corresponding tab. A click is one press and release of the mouse button.

2.1.6.1.2. Buttons



A button, also referred to as a push button, is used to perform a function indicated by the button label. To activate a button, place the pointer over the button and click the left mouse button.

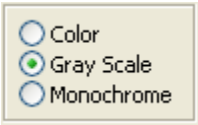
2.1.6.1.3. Check Boxes



A check box, also referred to as a check button, is used to enable / disable an item or action indicated by the check box label. Click the left mouse button on the check box to toggle the state.

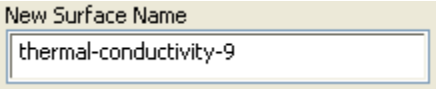
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.6.1.4. Radio Buttons



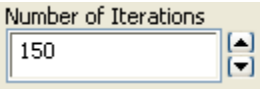
Radio buttons are a set of check boxes with the condition that only one can be set in the "on" position at a time. When you click the left mouse button on a radio button, it will be turned on, and all others will be turned off. Radio buttons appear either as diamonds (in Linux/UNIX systems) or as circles (as shown above).

2.1.6.1.5. Text Entry Boxes



A text entry box lets you type text input. It will often have a label associated with it to indicate the purpose of the entry.

2.1.6.1.6. Integer Number Entry Boxes

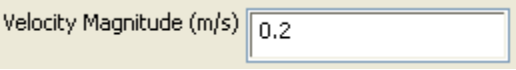


An integer number entry box is similar to a text entry except it allows only integer numbers to be entered (e.g., 10, -10, 50000 and 5E4). You may find it easier to enter large integer numbers using scientific notation. For example, you could enter `350000` or `3.5E5`.

The integer number entry also has arrow buttons that allow you to easily increase or decrease its value. For most integer number entry controls, the value will be increased (or decreased) by one when you click an arrow button. You can increase the size of the increment by holding down a keyboard key while clicking the arrow button. The keys used are shown below:

Key	Factor of Increase
`Shift`	`10`
`Ctrl`	`100`

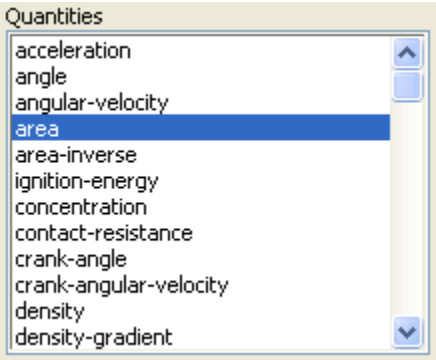
2.1.6.1.7. Real Number Entry Boxes



A real number entry box is similar to a text entry, except it allows only real numbers to be entered (e.g., 10, -10.538, 50000.45 and 5.72E-4). In most cases, the label will show the units associated with the real number entry.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

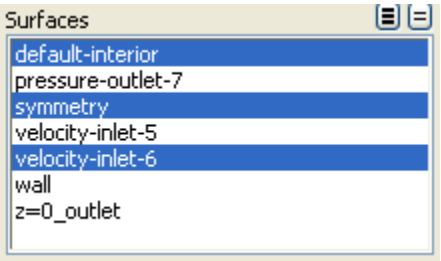
2.1.6.1.8. Single-Selection Lists



A single-selection list presents a list of items, with each item printed on a separate line. You can select an item by placing the pointer over the item line and clicking with the left mouse button. The selected item will become highlighted. Selecting another item will deselect the previously selected item in the list.

Many dialog boxes will also accept a double-click in order to invoke the dialog box action that is associated with the list selection (see information on the dialog box of interest for more details).

2.1.6.1.9. Multiple-Selection Lists

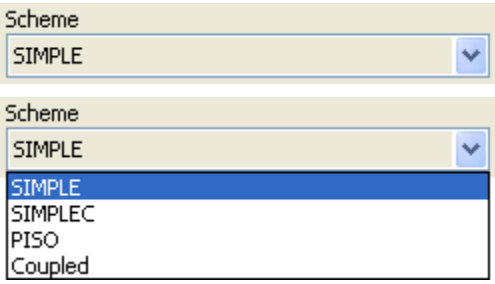


A multiple-selection list is similar to a single-selection list, except it allows for more than one selected item at a time. When you click the left mouse button on an item, its selection state will toggle. Clicking on an unselected item will select it. Clicking on a selected item will deselect it.

To select a range of items in a multiple-selection list, you can select the first desired item, and then select the last desired item while holding down the `Shift` key. The first and last items, and all the items between them, will be selected. You can also click and drag the left mouse button to select multiple items.

There are two small buttons in the upper right corner of the multiple selection list that accelerate the task of selecting or deselecting all the items in the list. Clicking the left button will select all items. Clicking the right button will deselect all items.

2.1.6.1.10. Drop-Down Lists



A drop-down list is a hidden single-selection list that shows only the current selection to save space.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you want to change the selection, follow the steps below:

1. Click the arrow button to display the list.
2. Place the pointer over the new list item.
3. Click the left mouse button on the item to make the selection and close the list.

If you wish to abort the selection operation while the list is displayed, you can move the pointer anywhere outside the list and click the left mouse button.

2.1.6.1.11. Scales

A scale is used to select a value from a predefined range by moving a slider. The number shows the current value. You can change the value by clicking the arrow buttons, or by following one of the procedures below:

1. Place the pointer over the slider.
2. Press and hold down the left mouse button.
3. Move the pointer along the slider bar to change the value.
4. Release the left mouse button.

or

1. Place the pointer over the slider and click the left mouse button.
2. Using the arrow keys on the keyboard, move the slider bar left or right to change the value.

2.1.6.1.11.1. Types of Dialog Boxes

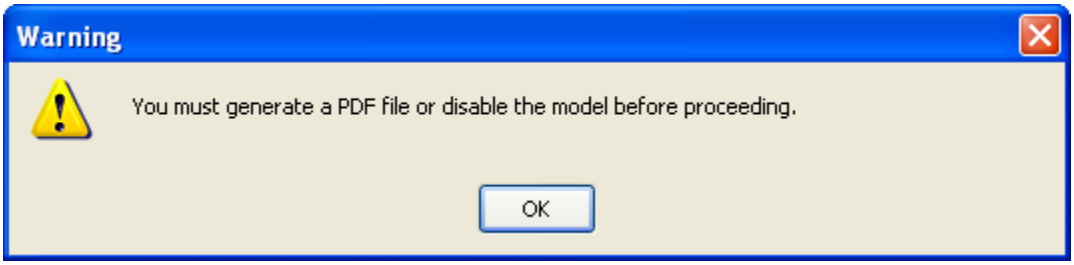
The following sections describe the various types of dialog boxes.

2.1.6.1.12. Information Dialog Boxes

The **Information** dialog box is used to report some information that ANSYS FLUENT thinks you should know. After you have read the information, you can click the **OK** button to close the dialog box.

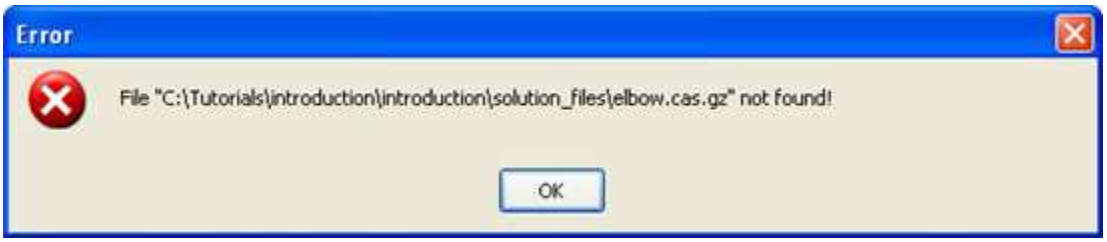
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.6.1.13. Warning Dialog Boxes



The **Warning** dialog box is used to warn you of a potential problem or deliver an important message. Your control of ANSYS FLUENT will be suspended until you acknowledge the warning by clicking the **OK** button.

2.1.6.1.14. Error Dialog Boxes

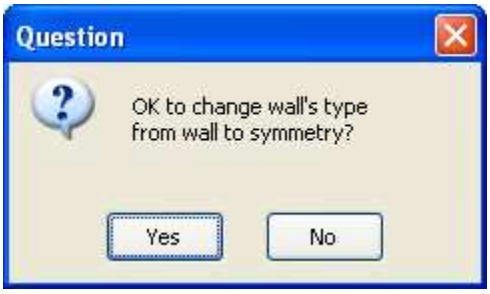


The **Error** dialog box is used to alert you of an error that has occurred. After you have read the error information, you can click the **OK** button to close the dialog box.

2.1.6.1.15. The Working Dialog Box

The **Working** dialog box is displayed when ANSYS FLUENT is busy performing a task. This is a special dialog box, because it requires no action by you. It is there to let you know that you must wait. When the program is finished, it will close the dialog box automatically. You can, however, abort the task that is being performed by clicking the **Cancel** button.

2.1.6.1.16. Question Dialog Box



The **Question** dialog box is used to ask you a question. Sometimes the question will require a **Yes** or **No** answer, while other times it will require that you either allow an action to proceed (**OK**) or **Cancel** the action. You can click the appropriate button to answer the question.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

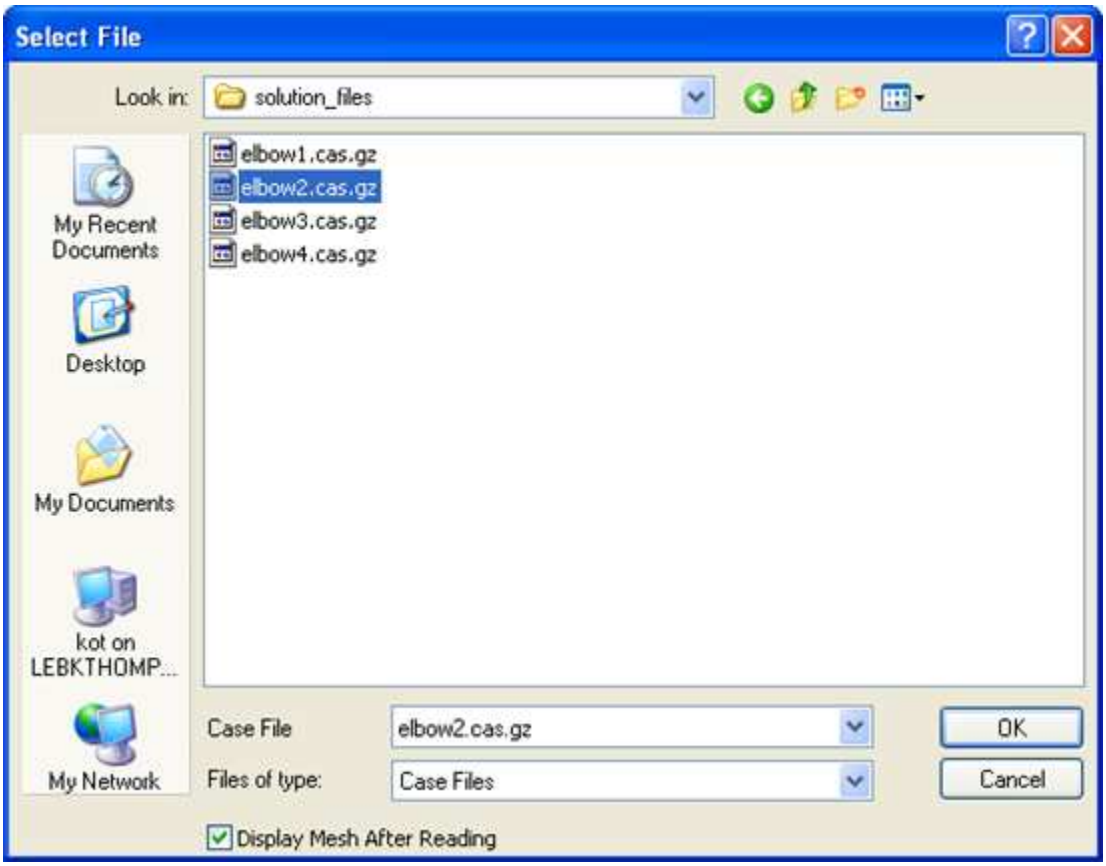
2.1.6.1.17. The Select File Dialog Box

File selection is accomplished using the **Select File** dialog box (*The **Select File** Dialog Box (Windows)* (p. 33) or *The Select File Dialog Box (UNIX or Linux)* (p. 33)).

2.1.6.1.17.1. The Select File Dialog Box (Windows)

File selection on Windows systems is accomplished using the standard Windows **Select File** dialog box (*Figure 2.6* (p. 33)).

Figure 2.6 The Select File Dialog Box for Windows



See documentation regarding your Windows system for further instructions on file selection.

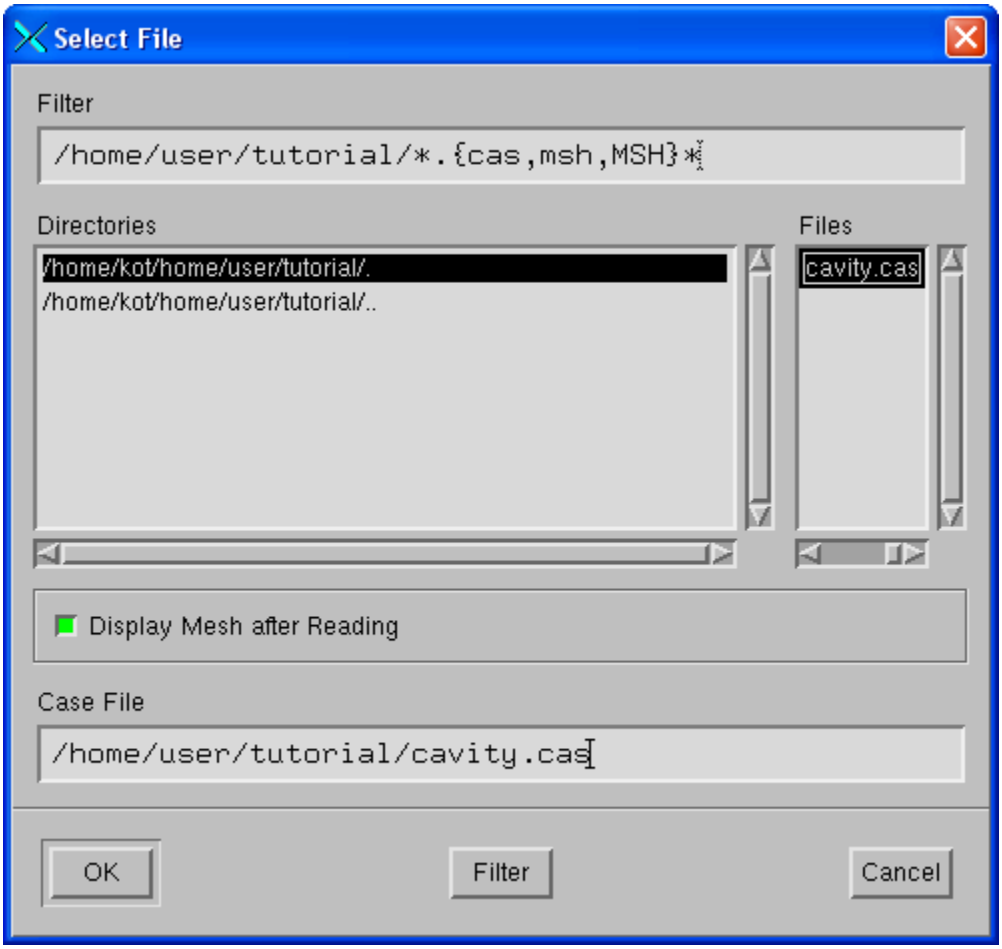
2.1.6.1.17.2. The Select File Dialog Box (UNIX or Linux)

For UNIX or Linux systems, note that the appearance of the **Select File** dialog box will not always be the same.

The version shown in *Figure 2.7* (p. 34) will appear in almost all cases, but it will be different if you are loading external data files for use in an XY plot (see *Including External Data in the Solution XY Plot* (p. 1472) for more information). In such cases, the dialog box will look like *Figure 2.8* (p. 35).

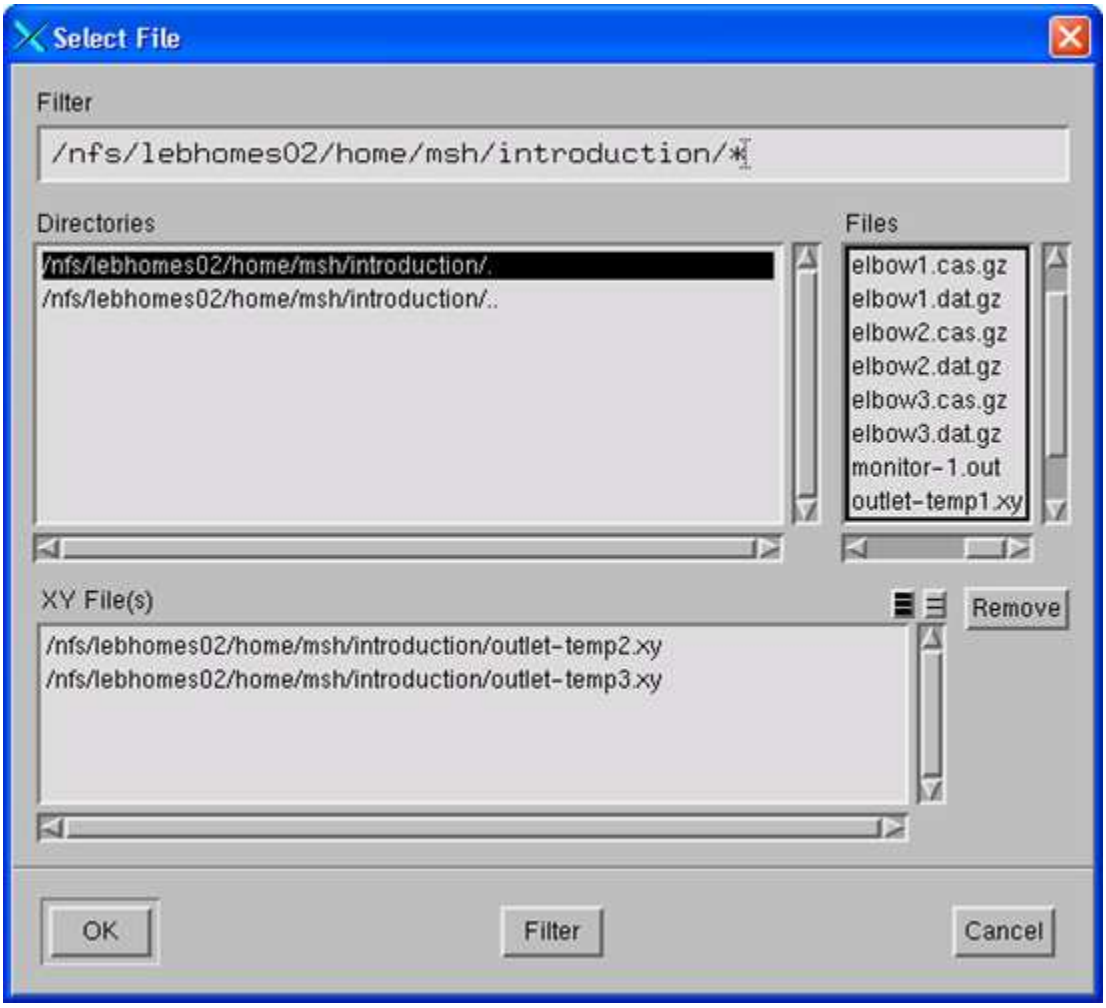
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.7 The Select File Dialog Box for UNIX or Linux Platforms



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.8 Another Version of the Select File Dialog Box for UNIX or Linux Platforms



The steps for file selection are as follows:

1. Go to the appropriate directory. You can do this in two different ways:
   - Enter the path to the desired directory in the **Filter** text entry box and then press the `<Enter>` key or click the **Filter** button. Be sure to include the final `/` character in the pathname, before the optional search pattern (described below).
   - Double-click a directory, and then a subdirectory, etc. in the **Directories** list until you reach the directory you want. You can also click once on a directory and then click the **Filter** button, instead of double-clicking. Note that the " **.**" item represents the current directory and the " **..**" item represents the parent directory.
2. Specify the file name by selecting it in the **Files** list or entering it in the **File** text entry box (if available) at the bottom of the dialog box. The name of this text entry box will change depending on the type of file you are selecting (**Case File**, **Journal File**, etc.).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that if you are searching for an existing file with a nonstandard extension, you may need to modify the "search pattern" at the end of the path in the **Filter** text entry box. For example, if you are reading a data file, the default extension in the search path will be `*.dat*`, and only those files that have a `.dat` extension will appear in the **Files** list. If you want files with a `.DAT` extension to appear in the **Files** list, you can change the search pattern to `*.DAT*`. If you want all files in the directory to be listed in the **Files** list, enter just `*` as the search pattern.

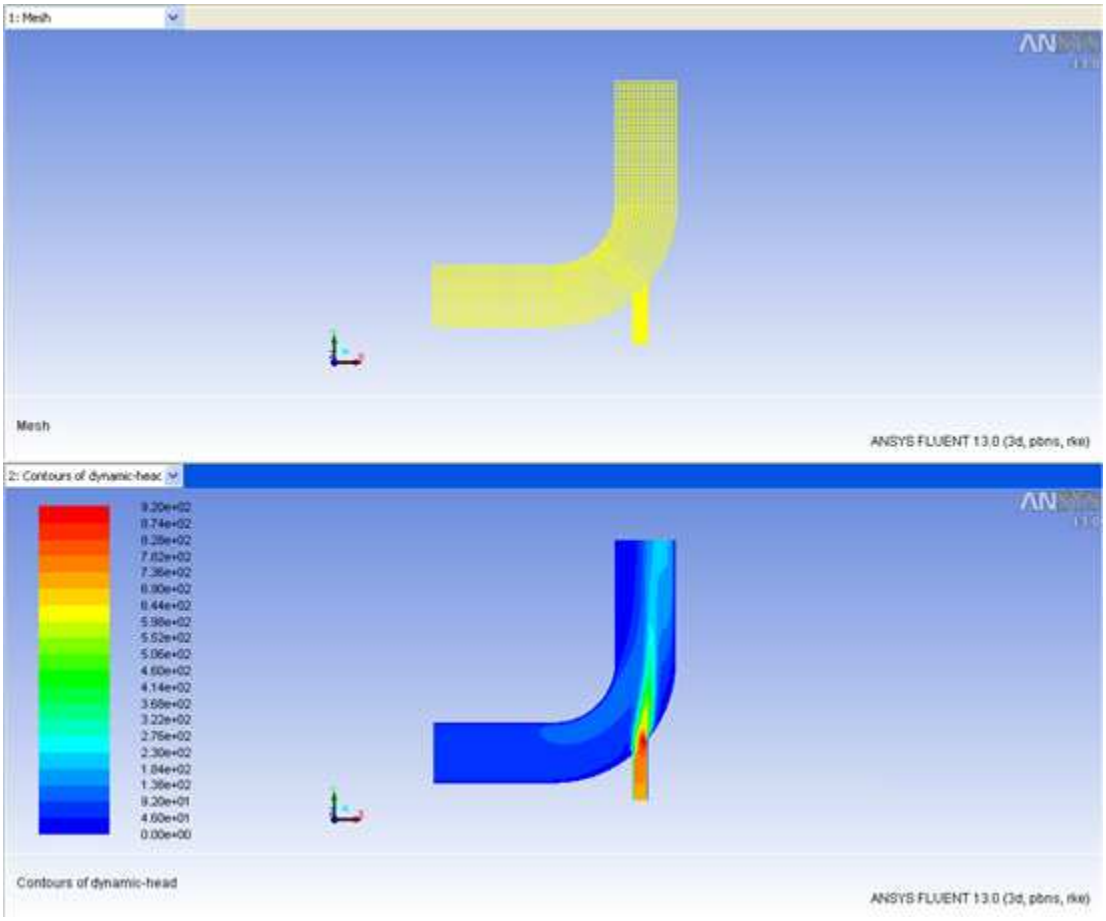
3. If you are reading a mesh or case file, use the **Display Mesh after Reading** option to specify whether you want ANSYS FLUENT to automatically display the mesh after the file is read. All of the boundary zones will be displayed, except for the interior zones of 3D geometries. The default status of this option (i.e., enabled or disabled) is determined by your decision regarding the **Display Mesh After Reading** option in FLUENT Launcher.
4. If you using the **Mesh/Replace...** menu item to replace a mesh for which data exists, you can enable the **Interpolate Data Across Zones** option to interpolate the data across cell zones. This option is appropriate when the matching zone pairs (i.e., the zones with the same names in both the current mesh and the replacement mesh) do not have the same interior zone boundaries. See *Replacing the Mesh* (p. 198) for details.
5. If you are reading multiple XY-plot data files, the selected file will be added to the list of **XY File(s)**. You can choose another file, following the instructions above, and it will also be added to this list. (If you accidentally select the wrong file, you can choose it in the **XY File(s)** list and click the **Remove** button to remove it from the list of files to be read.) Repeat until all of the desired files are in the **XY File(s)** list.
6. If you are writing a case, data, or radiation file, use the **Write Binary Files** check box to specify whether the file should be written as a text or binary file. You can read and edit a text file, but it will require more storage space than the same file in binary format. Binary files take up less space and can be read and written by ANSYS FLUENT more quickly.
7. Click the **OK** button to read or write the specified file. Shortcuts for this step are as follows:
   - If your file appears in the **Files** list *and* you are *not* reading an XY file, double-click it instead of just selecting it. This will automatically activate the **OK** button. (If you are reading an XY file, you will always have to click **OK** yourself. Clicking or double-clicking will just add the selected file to the **XY File(s)** list.)
   - If you entered the name of the file in the **File** text entry box, you can press the `<Enter>` key instead of clicking the **OK** button.

2.1.7. Graphics Windows

Graphics windows display the program's graphical output, and may be viewed within the ANSYS FLUENT application window or in separate windows. The decision to embed the graphics window or to have floating graphics windows is made when you start ANSYS FLUENT using FLUENT Launcher. For information about FLUENT Launcher, refer to *Setting General Options in FLUENT Launcher* (p. 3). When viewed within the application window, the graphics windows will be placed below the toolbar on the right, as shown in *Figure 2.1* (p. 21).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.9 Displaying Two Graphics Windows



In *Figure 2.9* (p. 37), two graphics windows are displayed by selecting the menu item **View/Graphics Window Layout**, then selecting . Although this setup only allows two windows to be visible at a given time, any number of graphics windows can be created. You can select any existing graphics window to be displayed in either location through the drop-down menu, which appears in the top left corner of the embedded graphics window. This drop-down displays the title of the window.

Important

You can change the text that appears in the graphics window title section by simply clicking in the area and editing it as you would in a text editor.

The **Display Options** dialog box (see *Display Options Dialog Box* (p. 1992)) can be used to change the attributes of the graphics window or to open another graphics window. The **Mouse Buttons** dialog box (see *Mouse Buttons Dialog Box* (p. 2157)) can be used to set the action taken when a mouse button is pressed in the graphics window.

Important

To cancel a display operation, press the `<Ctrl>` and the `c` keys together while the data is being processed in preparation for graphical display. You cannot cancel the operation after the program begins to draw in the graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Windows systems, there are special features for printing the contents of the graphics window directly. These features are not available on UNIX systems.

2.1.7.1. Printing the Contents of the Graphics Window (Windows Systems Only)

If you are using the Windows version of ANSYS FLUENT with free floating graphics windows (i.e., they are not embedded in the application window), you can display the graphics window's system menu by clicking in the upper-left corner of the graphics window. This menu contains the usual system commands, such as move, size, and close. Along with the system commands, ANSYS FLUENT includes three commands in the menu for printer and clipboard support. These commands are described below:

Page Setup...
: displays the **Page Setup** dialog box (see *Using the **Page Setup** Dialog Box (Windows Systems Only)* (p. 38)), which allows you to change attributes of the picture copied to the clipboard, or sent to a printer. Further details about this dialog box are included in the following section.

Print...
: displays the Microsoft Windows **Print** dialog box (see *Printing the Contents of the Graphics Window (Windows Systems Only)* (p. 38)), which enables you to send a copy of the picture to a printer. Some attributes of the copied picture can be changed using the **Page Setup** dialog box (see *Using the **Page Setup** Dialog Box (Windows Systems Only)* (p. 38)). Still more attributes of the final print can be specified within the Microsoft Windows **Print** and **Print Setup** dialog boxes (see documentation for Microsoft Windows and your printer for details).

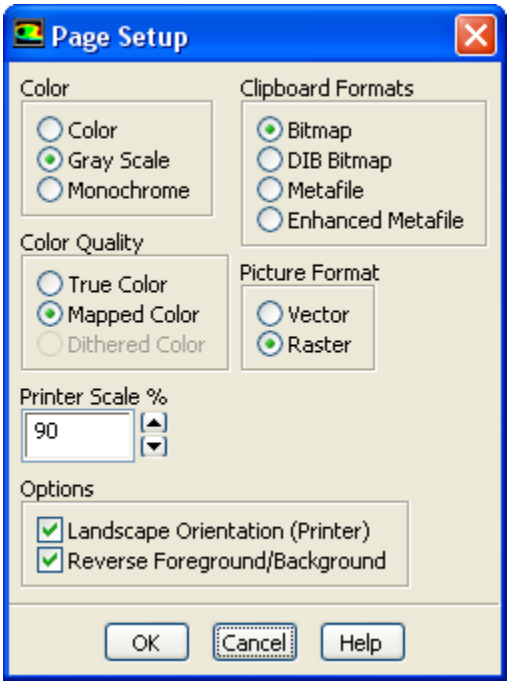
Copy to Clipboard
: places a copy of the current picture into the Microsoft Windows clipboard. Some attributes of the copied picture can be changed using the **Page Setup** dialog box (see *Using the **Page Setup** Dialog Box (Windows Systems Only)* (p. 38)). The size of your graphics window affects the size of the text fonts used in the picture. For best results, experiment with the graphics window size and examine the resulting clipboard picture using the Windows clipboard viewer.

2.1.7.2. Using the Page Setup Dialog Box (Windows Systems Only)

To open the **Page Setup** dialog box (see *Using the **Page Setup** Dialog Box (Windows Systems Only)* (p. 38)) (*Figure 2.10* (p. 39)), select the **Page Setup...** menu item in the system menu of the graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.10 The Page Setup Dialog Box (Windows Systems Only)



Controls

Color

allows you to specify a color or non-color picture.

Color

selects a color picture.

Gray Scale

selects a gray-scale picture.

Monochrome

selects a black-and-white picture.

Color Quality

allows you to specify the color mode used for the picture.

True Color

creates a picture defined by RGB values. This assumes that your printer or display has at least 65536 colors, or "unlimited colors" .

Mapped Color

creates a picture that uses a colormap. This is the right choice for devices that have 256 colors.

Clipboard Formats

allows you to choose the desired format copied to the clipboard. The size of your graphics window can affect the size of the clipboard picture. For best results, experiment with the graphics window size and examine the resulting clipboard picture using the Windows clipboard viewer.

Bitmap

is a bitmap copy of the graphics window.

DIB Bitmap

is a device-independent bitmap copy of the graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Metafile
is a Windows Metafile.

Enhanced Metafile
is a Windows Enhanced Metafile.

Picture Format
allows you to specify a raster or a vector picture.

Vector
creates a vector picture. This format will have a higher resolution when printed, but some large 3D pictures may take a long time to print.

Raster
creates a raster picture. This format will have a lower resolution when printed, but some large 3D pictures may take much less time to print.

Printer Scale %
controls the amount of the page that the printed picture will cover. Decreasing the scaling will effectively increase the margin between the picture and the edge of the paper.

Options
contains options that control other attributes of the picture.

Landscape Orientation (Printer)
specifies the orientation of the picture. If selected, the picture is made in landscape mode; otherwise, it is made in portrait mode. This option is applicable only when printing.

Reverse Foreground/Background
specifies that the foreground and background colors of the picture will be swapped. This feature allows you to make a copy of the picture with a white background and a black foreground, while the graphics window is displayed with a black background and white foreground.

2.2. Customizing the Graphical User Interface (UNIX Systems Only)

On UNIX systems, you may wish to customize the graphical user interface by changing attributes such as text color, background color, and text fonts. The program will try to provide default text fonts that are satisfactory for your platform's display size, but in some cases customization may be necessary if the default text fonts make the GUI too small or too large on your display, or if the default colors are undesirable.

The GUI in ANSYS FLUENT is based on the **X Window System Toolkit** and **OSF/Motif**. The attributes of the GUI are represented by X Window "resources" . If you are unfamiliar with the **X Window System Resource Database**, please refer to any documentation you may have that describes how to use the **X Window System** or **OSF/Motif** applications.

The default X Window resource values for a medium resolution display are shown below:

```
!
! General resources
!Fluent*geometry:              +0-0
Fluent*fontList:               *-helvetica-bold-r-normal--12-*
Fluent*MenuBar*fontList:       *-helvetica-bold-r-normal--12-*
Fluent*XmText*fontList:        *-fixed-medium-r-normal--13-*
Fluent*XmTextField*fontList:   *-fixed-medium-r-normal--13-*
Fluent*foreground:             black
Fluent*background:             gray75
Fluent*activeForeground:       black
Fluent*activeBackground:       gray85
Fluent*disabledTextColor:      gray55
Fluent*XmToggleButton.selectColor:       green
Fluent*XmToggleButtonGadget.selectColor: green
Fluent*XmText.translations:\
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        #override<Key>Delete: delete-previous-character()
Fluent*XmTextField.translations:\
        #override<Key>Delete: delete-previous-character()
!
! Console resources
!
Fluent*ConsoleText.rows:         24
Fluent*ConsoleText.columns:      80
Fluent*ConsoleText.background:   linen
!
! Help Viewer resources
!
Fluent*Hyper.foreground:         black
Fluent*Hyper.background:         linen
Fluent*Hyper.hyperColor:         SlateBlue3
Fluent*Hyper*normalFont:\
        *-new century schoolbook-medium-r-normal--12-*
Fluent*Hyper*hyperFont:\
        *-new century schoolbook-bold-r-normal--12-*
Fluent*Hyper*texLargeFont:\
        *-new century schoolbook-bold-r-normal--14-*
Fluent*Hyper*texBoldFont:\
        *-new century schoolbook-bold-r-normal--12-*
Fluent*Hyper*texFixedFont:\
        *-courier-bold-r-normal--12-*
Fluent*Hyper*texItalicFont:\
        *-new century schoolbook-medium-i-normal--12-*
Fluent*Hyper*texMathFont:\
        *-symbol-medium-r-normal--14-*
Fluent*Hyper*texSansFont:\
        *-helvetica-bold-r-normal--12-*
```

To customize one or more of the resources for a particular user, place appropriate resource specification lines in that user's file `$HOME/.Xdefaults` or whatever resource file is loaded by the X Window System on the user's platform.

To customize one or more of the resources for several users at a site, place the resource specification lines in an application defaults resource file called `Fluent`. This file should then be installed in a directory such as `/usr/lib/X11/app-defaults`, or on **SUN** workstations, the directory may be `/usr/open-win/lib/app-defaults`. See documentation regarding your platform for more information.

2.3. Using the GUI Help System

ANSYS FLUENT includes an integrated help system that provides easy access to the program documentation. Through the graphical user interface, you have the entire User's Guide and other documentation available to you with the click of a mouse button. The User's Guide and other manuals are displayed in the help viewer, which allows you to use hypertext links and the browser's search and index tools to find the information you need.

There are many ways to access the information contained in the online help. You can get reference information from within a task page or dialog box, or (on UNIX machines) request context-sensitive help for a particular menu item or dialog box. You can also go to the User's Guide contents page or the viewer index, and use the hypertext links to find the information you are looking for. In addition to the User's Guide, you can also access the other ANSYS FLUENT documentation (e.g., the Tutorial Guide or UDF Manual).

Note that the Reference Guide, which is the last chapter of the User's Guide in the online help, contains a description of each task page, menu item, and dialog box.

The sections that follow provide information on how to get help for a task page or dialog box, and brief descriptions of the **Help** menu items in ANSYS FLUENT. For more information, please refer to the information available from the **Help** menu in the viewer itself.

2.3.1. Task Page and Dialog Box Help

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.2. Context-Sensitive Help (UNIX Only)
2.3.3. Opening the User's Guide Table of Contents
2.3.4. Opening the User's Guide Index
2.3.5. Opening the Reference Guide
2.3.6. Help on Help
2.3.7. Accessing Printable Manuals
2.3.8. Help for Text Interface Commands
2.3.9. Accessing the Online Technical Support Web Site
2.3.10. Obtaining License Use Information
2.3.11. Version and Release Information

2.3.1. Task Page and Dialog Box Help

To get help about a task page or dialog box that you are currently using, click the **Help** button in the task page or dialog box. The help viewer will open to the section of the Reference Guide that explains the function of each item in the task page or dialog box. In this section you will also find hypertext links to the section(s) of the User's Guide that discuss how to use the task page or dialog box and provide related information.

2.3.2. Context-Sensitive Help (UNIX Only)

If you want to find out how or when a particular menu item or dialog box is used, you can use the context-sensitive help feature. Select the **Context-Sensitive Help** item in the **Help** pull-down menu.

Help → **Context-Sensitive Help**

With the resulting question-mark cursor, select an item from a pull-down menu. The help viewer will open to the section of the User's Guide that discusses the selected item.

2.3.3. Opening the User's Guide Table of Contents

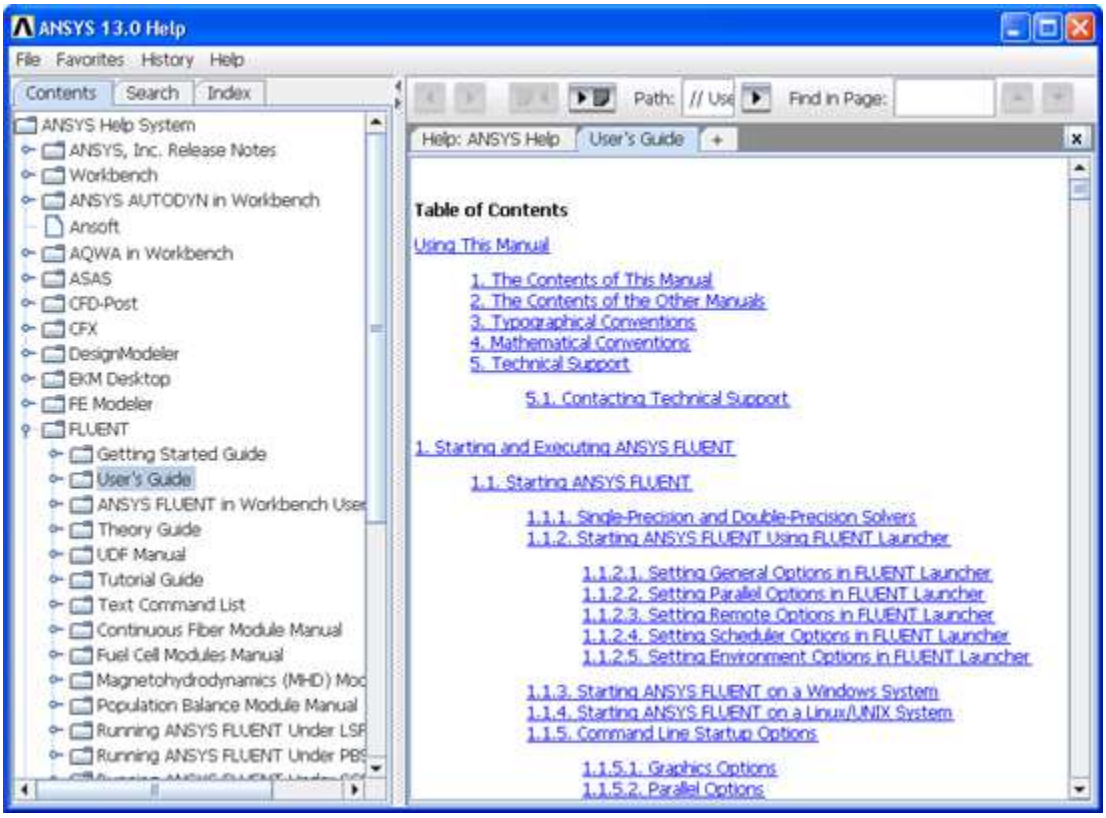
To see the User's Guide table of contents, select the **User's Guide Contents...** menu item in the **Help** pull-down menu.

Help → **User's Guide Contents...**

Selecting this item will open the help viewer to the contents page of the User's Guide (*Figure 2.11* (p. 43)). Each item in the table of contents is a hypertext link that you can click to view that chapter or section.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.11 The ANSYS FLUENT User's Guide Contents Page



In addition, the FLUENT documents are listed in the **Contents** tab of the help viewer. These listings can be expanded by clicking the icon to the left of the document title. The expanded list will contain hyperlinks to particular chapters or sections, and may also be expandable. For more information on navigation within the help viewer, please refer to the information available from the **Help** menu in the viewer itself.

2.3.4. Opening the User's Guide Index

Help → **User's Guide Index...**

The help viewer contains an index containing entries for most of the ANSYS product documentation. The **Index** tab allows you to search index terms and display the associated help topics.

To find a topic using the index, type the first few letters of a keyword in the field at the top of the **Index** tab.

2.3.5. Opening the Reference Guide

To open the help system to the first page of the Reference Guide, which contains information about each dialog box or menu item, arranged by pull-down menu, do the following in the help viewer:

1. Expand the FLUENT **User's Guide** in the left pane of the viewer by clicking the icon to the left of the document title.
2. Scroll down to the **Task Page Reference Guide** or **Menu Reference Guide** item in the left pane of the viewer and click the one of interest.
3. Use the hyperlinks in the main viewer window to find the topic of interest, or, expand the item in the left pane of the viewer and scroll to the topic of interest.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.6. Help on Help

Help → **Using Help...**

When you select this item, the help viewer will open to the beginning of this section.

You can obtain information about using online help by selecting the **Using Help...** menu item in the **Help** pull-down menu in the help viewer itself.

2.3.7. Accessing Printable Manuals

As noted above, you can access other manuals (in addition to the User's Guide) through the online help. You can also access the User's Guide and other manuals in printable format (PDF). To see ANSYS FLUENT manuals available in PDF, select the **PDF** menu item in the FLUENT **Help** pull-down menu.

Help → **PDF**

When you select this item, a list of documents will be shown in a submenu. Click the appropriate item to open that document with Acrobat Reader (version 5.0 or higher).

If you do not have Acrobat Reader, you can download it for free from Adobe (www.adobe.com).

See the Getting Started Guide for more information.

2.3.8. Help for Text Interface Commands

There are two ways to find information about text interface commands. You can either go to the **Text Command List** (which can be accessed using the **Help/More Documentation...** menu item, as described below), or use the text interface help system described in *Using the Text Interface Help System* (p. 55).

2.3.9. Accessing the Online Technical Support Web Site

You can access the Online Technical Support web site by selecting the **Online Technical Support...** menu item in the **Help** pull-down menu.

Help → **Online Technical Support...**

ANSYS FLUENT will direct your web browser to the appropriate web address.

2.3.10. Obtaining License Use Information

If you are running with an existing ANSYS FLUENT license (FluentLM) you can obtain a listing of current ANSYS FLUENT users when you select the **License Usage...** menu item in the **Help** pull-down menu.

Help → **License Usage...**

ANSYS FLUENT will display a list of the current users of the ANSYS FLUENT license feature in the console.

If your installation of ANSYS FLUENT is managed by the ANSYS License Manager (ANSLIC_ADMIN), you will see a message that will indicate that licensing is managed by ANSLIC_ADMIN. For additional information on licensing information, please refer to the **Installation and Licensing Documentation** within the help viewer. This information can be found by doing the following in the help viewer:

1. Scroll down to the **Installation and Licensing Documentation** item in the left pane of the vlewer.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Expand this document by clicking on the icon to the left of the document title.
3. Use the hyperlinks in the main viewer window to find the desired information, or, expand the items in the left pane of the viewer and scroll to the topic of interest.

2.3.11. Version and Release Information

You can obtain information about the version and release of ANSYS FLUENT you are running by selecting the **Version...** menu item in the **Help** pull-down menu.

Help → **Version...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Text User Interface (TUI)

In addition to the graphical user interface described in *Graphical User Interface (GUI)* (p. 21), the user interface in ANSYS FLUENT includes a textual command line interface.

3.1. Text Menu System
3.2. Text Prompt System
3.3. Interrupts
3.4. System Commands
3.5. Text Menu Input from Character Strings
3.6. Using the Text Interface Help System

The text interface (TUI) uses, and is written in, a dialect of Lisp called Scheme. Users familiar with Scheme will be able to use the interpretive capabilities of the interface to create customized commands.

3.1. Text Menu System

The text menu system provides a hierarchical interface to the program's underlying procedural interface. Because it is text based, you can easily manipulate its operation with standard text-based tools: input can be saved in files, modified with text editors, and read back in to be executed. Because the text menu system is tightly integrated with the Scheme extension language, it can easily be programmed to provide sophisticated control and customized functionality.

The menu system structure is similar to the directory tree structure of UNIX operating systems. When you first start ANSYS FLUENT, you are in the "root" menu and the menu prompt is a greater-than symbol.

```
>
```

To generate a listing of the submenus and commands in the current menu, simply press `<Enter>`.

```
><Enter>
adapt/       file/         report/
define/      mesh/         solve/
display/     parallel/     surface/
exit         plot/         view/
```

By convention, submenu names end with a `/` to differentiate them from menu commands. To execute a command, just type its name (or an abbreviation). Similarly, to move down into a submenu, enter its name or an abbreviation. When you move into the submenu, the prompt will change to reflect the current menu name.

```
>display

/display> set

/display/set>
```

To move back to the previously occupied menu, type `q` or `quit` at the prompt.

```
/display/set> q

/display>
```

You can move directly to a menu by giving its full pathname.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/display /file

/display//file>
```

In the above example, control was passed from `/display` to `/file` without stopping in the root menu. Therefore, when you quit from the `/file` menu, control will be passed directly back to `/display`.

```
/display//file> q

/display>
```

Furthermore, if you execute a command without stopping in any of the menus along the way, control will again be returned to the menu from which you invoked the command.

```
/display> /file start-journal jrnl

 Opening input journal to file "jrnl".

/display>
```

The text menu system provides on-line help for menu commands. The text menu on-line help system is described in *Using the Text Interface Help System* (p. 55).

To edit the current command, you can position the cursor with the left and right arrow keys, delete with the `<Backspace>` key, and insert text simply by typing.

For additional information, please see the following sections:

3.1.1. Command Abbreviation
3.1.2. Command Line History
3.1.3. Scheme Evaluation
3.1.4. Aliases

3.1.1. Command Abbreviation

To select a menu command, you do not need to type the entire name; you can type an abbreviation that matches the command. The rules for matching a command are as follows: A command name consists of phrases separated by hyphens. A command is matched by matching an initial sequence of its phrases. Matching of hyphens is optional. A phrase is matched by matching an initial sequence of its characters. A character is matched by typing that character.

If an abbreviation matches more than one command, then the command with the greatest number of matched phrases is chosen. If more than one command has the same number of matched phrases, then the first command to appear in the menu is chosen.

For example, each of the following will match the command `set-ambient-color`: `set- ambient-color`, `s-a-c`, `sac`, and `sa`. When abbreviating commands, sometimes your abbreviation will match more than one command. In such cases, the first command is selected. Occasionally, there is an anomaly such as `lint` not matching `lighting-interpolation` because the `li` gets absorbed in `lights-on`? and then the `nt` does not match `interpolation`. This can be resolved by choosing a different abbreviation, such as `liin`, or `l-int`.

3.1.2. Command Line History

You can use the up and down arrow keys on your keyboard to go through recently used commands that are stored in history. By default, command-history will store only the last ten commands. This can be changed (for example to 15) by using the following command:

```
> (set! *cmd-history-length* 15)
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Command-history is not available if the ANSYS FLUENT application is started with -g options (see *Command Line Startup Options* (p. 10)).

Important

The user inputs supplied as the arguments of the TUI command or alias will not be saved in history. By way of illustration, consider the following entry in the TUI:

```
> rc new_file.cas
```

Important

In history, only `rc` (an alias for `read-case`) will be saved, since `new_file.cas` is a user input to the alias-function.

Commands recalled from history can be edited or corrected using the `<Backspace>` key and the left and right arrow keys.

3.1.3. Scheme Evaluation

If you enter an open parenthesis, `(`, at the menu prompt, then that parenthesis and all characters up to and including the matching closing parenthesis are passed to Scheme to be evaluated, and the result of evaluating the expression is displayed.

```
> (define a 1)
a
> (+ a 2 3 4)
10
```

3.1.4. Aliases

Command aliases can be defined within the menu system. As with the UNIX `csh` shell, aliases take precedence over command execution. The following aliases are predefined in Cortex: `error`, pwd, `chdir`, `ls`, `.`, and `alias`.

`error`
displays the Scheme object that was the "irritant" in the most recent Scheme error interrupt.

`pwd`
prints the working directory in which all file operations will take place.

`chdir`
will change the working directory.

`ls`
lists the files in the working directory.

`alias`
displays the list of symbols currently aliased.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2. Text Prompt System

Commands require various arguments, including numbers, filenames, yes/no responses, character strings, and lists. A uniform interface to this input is provided by the text prompt system. A prompt consists of a prompt string, followed by an optional units string enclosed in parentheses, followed by a default value enclosed in square brackets. The following shows some examples of prompts:

```
filled-mesh? [no] <Enter>

shrink-factor [0.1] <Enter>

line-weight [1] <Enter>

title ["] <Enter>
```

The default value for a prompt is accepted by pressing `<Enter>` on the keyboard or typing a `,` (comma).

Important

Note that a comma is not a separator. It is a separate token that indicates a default value. The sequence "`1,2`" results in three values; the number 1 for the first prompt, the default value for the second prompt, and the number 2 for the third prompt.

A short help message can be displayed at any prompt by entering a `?`. (See *Using the Text Interface Help System* (p. 55).)

To abort a prompt sequence, simply press `<Ctrl>` and the `c` keys together.

For additional information, please see the following sections:

3.2.1. Numbers
3.2.2. Booleans
3.2.3. Strings
3.2.4. Symbols
3.2.5. Filenames
3.2.6. Lists
3.2.7. Evaluation
3.2.8. Default Value Binding

3.2.1. Numbers

The most common prompt type is a number. Numbers can be either integers or reals. Valid numbers are, for example, `16`, `-2.4`, `.9E5`, and `+1E-5`. Integers can also be specified in binary, octal, and hexadecimal form. The decimal integer `31` can be entered as `31`, `#b11111`, `#o37`, or `#x1f`. In Scheme, integers are a subset of reals, so you do not need a decimal point to indicate that a number is real; `2` is just as much a real as `2.0`. If you enter a real number at an integer prompt, any fractional part will simply be truncated; `1.9` will become `1`.

3.2.2. Booleans

Some prompts require a yes-or-no response. A yes/no prompt will accept either `yes` or `y` for a positive response, and `no` or `n` for a negative response. yes/no prompts are used for confirming potentially dangerous actions such as overwriting an existing file, exiting without saving case, data, mesh, etc.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Some prompts require actual Scheme boolean values (true or false). These are entered with the Scheme symbols for true and false, `#t` and `#f`.

3.2.3. Strings

Character strings are entered in double quotes, e.g., `"red"`. Plot titles and plot legend titles are examples of character strings. Character strings can include any characters, including blank spaces and punctuation.

3.2.4. Symbols

Symbols are entered *without* quotes. Zone names, surface names, and material names are examples of symbols. Symbols must start with an alphabetical character (i.e., a letter), and cannot include any blank spaces or commas.

3.2.5. Filenames

Filenames are actually just character strings. For convenience, filename prompts do not require the string to be surrounded with double quotes. If, for some exceptional reason, a filename contains an embedded space character, then the name must be surrounded with double quotes.

One consequence of this convenience is that filename prompts do not evaluate the response. For example, the sequence

```
> (define fn "valve.ps")

fn

> hc fn
```

will end up writing a picture file with the name `fn`, not `valve.ps`. Since the filename prompt did not evaluate the response, `fn` did not get a chance to evaluate `"valve.ps"` as it would for most other prompts.

3.2.6. Lists

Some functions in ANSYS FLUENT require a "list" of objects such as numbers, strings, booleans, etc. A list is a Scheme object that is simply a sequence of objects terminated by the empty list, `'()`. Lists are prompted for an element at a time, and the end of the list is signaled by entering an empty list. This terminating list forms the tail of the prompted list, and can either be empty or can contain values. For convenience, the empty list can be entered as `()` as well as the standard form `'()`. Normally, list prompts save the previous argument list as the default. To modify the list, overwrite the desired elements and terminate the process with an empty list. For example,

```
element(1) [()] 1

element(2) [()] 10

element(3) [()] 100

element(4) [()] <Enter>
```

creates a list of three numbers: 1, 10, and 100. Subsequently,

```
element(1) [1] <Enter>

element(2) [10] <Enter>

element(3) [100] <Enter>

element(4) [()] 1000
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
element(5) [()] <Enter>
```

adds a fourth element. Then

```
element(1) [1] <Enter>
element(2) [10] <Enter>
element(3) [100] ()
```

leaves only 1 and 10 in the list. Subsequently entering

```
element(1) [1] ,,'(11 12 13)
```

creates a five element list: 1, 10, 11, 12, and 13. Finally, a single empty list removes all elements

```
element(1) [1] ()
```

A different type of list, namely, a "list-of-scalars" contains pick menu items (and not list items) for which a selection has to be made from listed quantities, which are available at the `Enter` prompt. Hence, a list-of-scalars cannot be entered as a list.

An example of a "list-of-scalars" consists of the following:

```
ASCII scalar(1)>
abs-angular-coordinate  cell-warp               radial-velocity
absolute-pressure       cell-weight             rel-tangential-velocity
adaption-curvature      cell-zone               rel-total-pressure
adaption-function       custom-function-0       rel-velocity-magnitude
adaption-iso-value      density                 relative-velocity-angle
adaption-space-gradient density-all             relative-x-velocity
angular-coordinate      dp-dx                   relative-y-velocity
axial-coordinate        dp-dy                   relative-z-velocity
axial-velocity          dp-dz                   tangential-velocity
boundary-cell-dist      dynamic-pressure        total-pressure
boundary-normal-dist    existing-value          velocity-angle
boundary-volume-dist    face-area-magnitude     velocity-magnitude
cell-children           face-handedness         x-coordinate
cell-element-type       face-squish-index       x-face-area
cell-equiangle-skew     interface-wall-zone     x-velocity
cell-equivolume-skew    mass-imbalance          x-velocity-residual
cell-partition          mesh-x-velocity         y-coordinate
cell-refine-level       mesh-y-velocity         y-face-area
cell-squish-index       mesh-z-velocity         y-velocity
cell-surface-area       partition-neighbors     y-velocity-residual
cell-type               pressure                z-coordinate
cell-volume             pressure-coefficient    z-face-area
cell-volume-change      pressure-residual       z-velocity
cell-wall-distance      radial-coordinate       z-velocity-residual

ASCII scalar(1)> pressure

ASCII scalar(2)> z-coordinate

ASCII scalar(3)> cell-volume

ASCII scalar(4)> dynamic-pressure velocity-magnitude x-face-area

ASCII scalar(7)>
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.7. Evaluation

All responses to prompts (except filenames, see above) are evaluated by the Scheme interpreter before they are used. You can therefore enter any valid Scheme expression as the response to a prompt. For example, to enter a unit vector with one component equal to $1/3$ (without using your calculator),

```
/foo> set-xy
x-component [1.0] (/ 1 3)

y-component [0.0] (sqrt (/ 8 9))
```

or, you could first define a utility function to compute the second component of a unit vector,

```
> (define (unit-y x) (sqrt (- 1.0 (* x x))))

unit-y
/foo> set-xy

x-component [1.0] (/ 1 3)

y-component [0.0] (unit-y (/ 1 3))
```

3.2.8. Default Value Binding

The default value at any prompt is bound to the Scheme symbol "_" (underscore) so that the default value can form part of a Scheme expression. For example, if you want to decrease a default value so that it is one-third of the original value, you could enter

```
shrink-factor [0.8] (/ _ 3)
```

3.3. Interrupts

The execution of the code can be halted by pressing the `<Ctrl>` and the `c` keys together, at which time the present operation stops at the next recoverable location.

3.4. System Commands

The way you execute system commands with the `!` (bang) shell escape character will be slightly different for UNIX and Windows systems.

For additional information, please see the following sections:

3.4.1. System Commands for UNIX-based Operating Systems
3.4.2. System Commands for Windows Operating Systems

3.4.1. System Commands for UNIX-based Operating Systems

If you are running ANSYS FLUENT under a UNIX-based operating system, all characters following the `!` up to the next newline character will be executed in a subshell. Any further input related to these system commands must be entered in the window in which you started the program, and any screen output will also appear in that window. (Note that if you started ANSYS FLUENT remotely, this input and output will be in the window in which you started Cortex.)

```
> !rm junk.*

> !vi script.rp
```

`!pwd` and `!ls` will execute the UNIX commands in the directory in which Cortex was started. The screen output will appear in the window in which you started ANSYS FLUENT, unless you started it remotely, in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which case the output will appear in the window in which you started Cortex. (Note that `!cd` executes in a subshell, so it will not change the working directory either for ANSYS FLUENT or for Cortex, and is therefore not useful.) Typing `cd` with no arguments will move you to your home directory in the console.

ANSYS FLUENT includes three system command aliases (`pwd`, `ls`, and `chdir`) that will be executed in your working directory with output displayed in the ANSYS FLUENT console. Note that these aliases will invoke the corresponding UNIX commands with respect to the parent directory of the case file. For example, `pwd` prints the parent directory of the case file in the ANSYS FLUENT console, while `!pwd` prints the directory from which you started ANSYS FLUENT in the UNIX shell window where you started ANSYS FLUENT.

Several examples of system commands entered in the console are shown below. The screen output that will appear in the window in which ANSYS FLUENT was started (or, if you started the program remotely, in the window in which Cortex was started) follows the examples.

Example input (in the ANSYS FLUENT console):

```
> !pwd

> !ls valve*.*
```

Example output (in the window in which ANSYS FLUENT— or Cortex, if you started the program remotely—was started):

```
/home/cfd/run/valve
 valve1.cas  valve1.msh  valve2.cas  valve2.msh
```

3.4.2. System Commands for Windows Operating Systems

If you are running ANSYS FLUENT under a Windows operating system, all characters following the `!` up to the next newline character will be executed. The results of a command will appear in the ANSYS FLUENT console, or in a separate window if the command starts an external program, such as Notepad.

```
> !del junk.*

> !notepad script.rp
```

`!cd` and `!dir` will execute the DOS commands and the screen output will appear in the ANSYS FLUENT console. The `!cd` command with no argument will display the current working directory in the ANSYS FLUENT console.

Several examples of system commands entered in the console are shown below.

Example input (in boxes) and output (in the ANSYS FLUENT console):

```
  > !cd

p:/cfd/run/valve
 > !dir valve*.*/w

     Volume in drive P is users
     Volume Serial Number is 1234-5678
     Directory of p:/cfd/run/valve
     valve1.cas      valve1.msh      valve2.cas      valve2.msh
        4 File(s)              621,183 bytes
        0 Dir(s)          1,830,088,704 bytes free
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.5. Text Menu Input from Character Strings

Often, when writing a Scheme extension function for ANSYS FLUENT, it is convenient to be able to include menu commands in the function. This can be done with `ti-menu-load-string`. For example, to open graphics window 2, use

```
(ti-menu-load-string "di ow 2")
```

A Scheme loop that will open windows 1 and 2 and display the front view of the mesh in window 1 and the back view in window 2 is given by

```
(for-each
 (lambda (window view)
   (ti-menu-load-string (format #f "di ow ~a gr view rv ~a"
 window view)))
 '(1 2)
 '(front back))
```

This loop makes use of the `format` function to construct the string used by `menu-load-string`. This simple loop could also be written without using menu commands at all, but you need to know the Scheme functions that get executed by the menu commands to do it:

```
(for-each
 (lambda (window view)
   (cx-open-window window)
   (display-mesh)
   (cx-restore-view view))
 '(1 2) '(front back))
```

String input can also provide an easy way to create aliases within ANSYS FLUENT. For example, to create an alias that will display the mesh, you could type the following:

```
(alias 'dg (lambda () (ti-menu-load-string "/di gr")))
```

Then any time you enter `dg` from anywhere in the menu hierarchy, the mesh will be drawn in the active window.

Important

`ti-menu-load-string` evaluates the string argument in the top level menu. It ignores any menu you may be in when you invoke `ti-menu-load-string`.

As a result, the command

```
(ti-menu-load-string "open-window 2 gr") ; incorrect usage
```

will not work even if you type it from within the `display/` menu—the string itself must cause control to enter the `display/` menu, as in

```
(ti-menu-load-string "display open-window 2 mesh")
```

3.6. Using the Text Interface Help System

The text user interface provides context-sensitive on-line help. Within the text menu system, you can obtain a brief description of each of the commands by entering a ? followed by the command in question.

Example:

```
> ?dis
display/: Enter the display menu.
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also enter a lone ? to enter "help mode." In this mode, you need only enter the command or menu name to display the help message. To exit the help mode type `q` or `quit` as for a normal menu.

Example:

```
  > ?

 [help-mode]> di

 display/: Enter the display menu.

  [help-mode]> pwd

  pwd: #[alias]
  (LAMBDA ()
    (BEGIN
    (SET! pwd-cmd ((LAMBDA n
   n) 'system (IF (cx-send '(unix?))
     "pwd"
     "cd")))
   (cx-send pwd-cmd)))

  [help-mode]> q
```

To access the help, type a ? at the prompt when you are prompted for information.

Example:

```
 > display/annotate Annotation text [""] ?
  Annotation text [""]
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Reading and Writing Files

During an ANSYS FLUENT session you may need to import and export several kinds of files. Files that are read include mesh, case, data, profile, Scheme, and journal files. Files that are written include case, data, profile, journal, and transcript files. ANSYS FLUENT also has features that allow you to save pictures of graphics windows. You can also export data for use with various visualization and postprocessing tools. These operations are described in the following sections.

4.1. Shortcuts for Reading and Writing Files
4.2. Reading Mesh Files
4.3. Reading and Writing Case and Data Files
4.4. Reading and Writing Parallel Data Files
4.5. Reading FLUENT/UNS and RAMPANT Case and Data Files
4.6. Reading and Writing Profile Files
4.7. Reading and Writing Boundary Conditions
4.8. Writing a Boundary Mesh
4.9. Reading Scheme Source Files
4.10. Creating and Reading Journal Files
4.11. Creating Transcript Files
4.12. Importing Files
4.13. Exporting Solution Data
4.14. Exporting Solution Data after a Calculation
4.15. Exporting Steady-State Particle History Data
4.16. Exporting Data During a Transient Calculation
4.17. Exporting to ANSYS CFD-Post
4.18. Managing Solution Files
4.19. Mesh-to-Mesh Solution Interpolation
4.20. Mapping Electromagnetic Losses
4.21. Mapping Data for Fluid-Structure Interaction (FSI) Applications
4.22. Saving Picture Files
4.23. Setting Data File Quantities
4.24. The .fluent File

4.1. Shortcuts for Reading and Writing Files

The following features in ANSYS FLUENT make reading and writing files convenient:

- Automatic appending or detection of default file name suffixes
- Binary file reading and writing
- Automatic detection of file format (text/binary)
- Recent file list
- Reading and writing of compressed files
- Tilde expansion
- Automatic numbering of files
- Ability to disable the overwrite confirmation prompt

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Standard toolbar buttons for reading and writing files

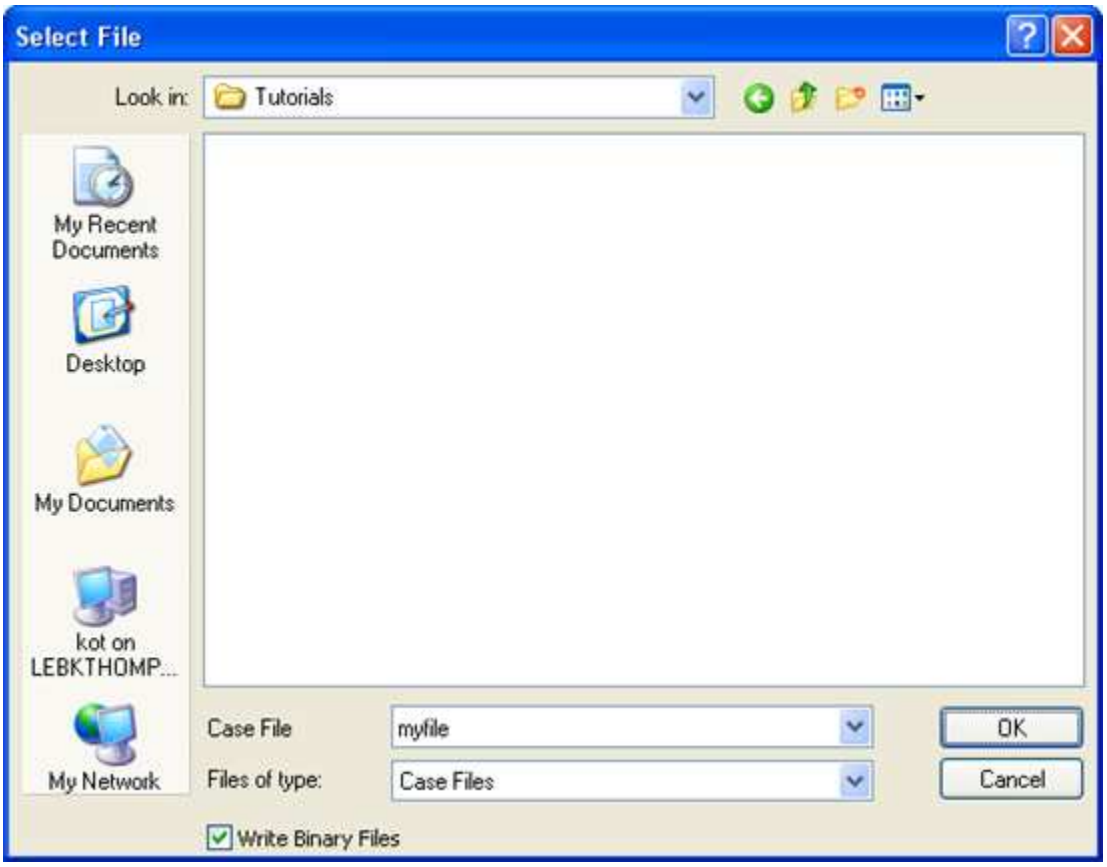
For additional information, please see the following sections:

4.1.1. Default File Suffixes
4.1.2. Binary Files
4.1.3. Detecting File Format
4.1.4. Recent File List
4.1.5. Reading and Writing Compressed Files
4.1.6. Tilde Expansion (Linux/UNIX Systems Only)
4.1.7. Automatic Numbering of Files
4.1.8. Disabling the Overwrite Confirmation Prompt
4.1.9. Toolbar Buttons

4.1.1. Default File Suffixes

Each type of file read or written in ANSYS FLUENT has a default file suffix associated with it. When you specify the first part of the file name (the prefix) for the commonly used files, the solver automatically appends or detects the appropriate suffix. For example, to write a case file named `myfile.cas`, just specify the prefix `myfile` in the **Select File** dialog box (*Figure 4.1* (p. 58)) and `.cas` is automatically appended. Similarly, to read the case file named `myfile.cas` into the solver, you can just specify `myfile` and ANSYS FLUENT automatically searches for a file of that name with the suffix `.cas`.

Figure 4.1 The Select File Dialog Box



The default file suffix for case and data files, PDF (Probability Density Function) files, DTRM ray files, profiles, scheme files, journal files, etc., are automatically detected and appended. The appropriate default file suffix appears in the **Select File** dialog box for each type of file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.2. Binary Files

When you write a case, data, or ray file, a binary file is saved by default. Binary files take up less memory than text files and can be read and written by ANSYS FLUENT more quickly.

Note

You cannot read and edit a binary file, as you can do for a text file.

To save a text file, turn off the **Write Binary Files** option in the **Select File** dialog box when you are writing the file.

4.1.3. Detecting File Format

When you read a case, data, mesh, PDF, or ray file, the solver automatically determines whether it is a text (formatted) or binary file.

4.1.4. Recent File List

At the bottom of the **File/Read** submenu there is a list of four ANSYS FLUENT case files that you most recently read or wrote. To read one of these files into ANSYS FLUENT, select it in the list. This allows you to read a recently used file without selecting it in the **Select File** dialog box.

Note that the files listed in this submenu may not be appropriate for your current session (e.g., a 3D case file can be listed even if you are running a 2D version of ANSYS FLUENT). Also, if you read a case file using this shortcut, the corresponding data file is read only if it has the same base name as the case file (e.g., `file1.cas` and `file1.dat`) *and* it was read/written with the case file the last time the case file was read/written.

4.1.5. Reading and Writing Compressed Files

4.1.5.1. Reading Compressed Files

You can use the **Select File** dialog box to read files compressed using `compress` or `gzip`. If you select a compressed file with a `.Z` extension, ANSYS FLUENT automatically invokes `zcat` to import the file. If you select a compressed file with a `.gz` extension, the solver invokes `gunzip` to import the file.

For example, if you select a file named `flow.msh.gz`, the solver reports the following message indicating that the result of the `gunzip` is imported into ANSYS FLUENT via an operating system pipe.

```
Reading "\"| gunzip -c \"Y:\flow.msh.gz\"\""...
```

You can also type in the file name without any suffix (e.g., if you are not sure whether or not the file is compressed). First, the solver attempts to open a file with the input name. If it cannot find a file with that name, it attempts to locate files with default suffixes and extensions appended to the name.

For example, if you enter the name `file-name`, the solver traverses the following list until it finds an existing file:

- `file-name`
- `file-name.gz`
- `file-name.Z`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `file-name.`*suffix*
- `file-name.`*suffix*`.gz`
- `file-name.`*suffix*`.Z`

where *suffix* is a common extension to the file, such as `.cas` or `.msh`. The solver reports an error if it fails to find an existing file with one of these names.

Important

- For Windows systems, only files that were compressed with `gzip` (i.e., files with a `.gz` extension) can be read. Files that were compressed with `compress` cannot be read into ANSYS FLUENT on a Windows machine.
- Do not read a compressed ray file; ANSYS FLUENT cannot access the ray tracing information properly from a compressed ray file.

4.1.5.2. Writing Compressed Files

You can use the **Select File** dialog box to write a compressed file by appending a `.Z` or `.gz` extension onto the file name.

For example, if you enter `flow.gz` as the name for a case file, the solver reports the following message:

```
Writing "| gzip -cfv > Y:\flow.cas.gz"...
```

The status message indicates that the case file information is being piped into the `gzip` command, and that the output of the compression command is being redirected to the file with the specified name. In this particular example, the `.cas` extension is added automatically.

Important

- For Windows systems, compression can be performed only with `gzip`. That is, you can write a compressed file by appending `.gz` to the name, but appending `.Z` does not compress the file.
- Do not write a compressed ray file; ANSYS FLUENT cannot access the ray tracing information properly from a compressed ray file.
- To compress parallel data files (i.e., files saved with a `.pdat` extension), you must use asynchronous file compression. (For details, see *Reading and Writing Case and Data Files* (p. 63)).

4.1.6. Tilde Expansion (Linux/UNIX Systems Only)

On Linux/UNIX systems, if you specify `~/` as the first two characters of a file name, the ~ is expanded as your home directory. Similarly, you can start a file name with `~username/`, and the `~username` is expanded to the home directory of "username". If you specify `~/file` as the case file to be written, ANSYS FLUENT saves the file `file.cas` in your home directory. You can specify a subdirectory of your home directory as well: if you enter `~/cases/file.cas`, ANSYS FLUENT saves the file `file.cas` in the `cases` subdirectory.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.7. Automatic Numbering of Files

There are several special characters that you can include in a file name. Using one of these character strings in your file name provides a shortcut for numbering the files based on various parameters (i.e., iteration number, time step, or total number of files saved so far), because you need not enter a new file name each time you save a file. (See also *Automatic Saving of Case and Data Files* (p. 65) for information about saving and numbering case and data files automatically.)

- For transient calculations, you can save files with names that reflect the time step at which they are saved by including the character string `%t` in the file name. For example, you can specify `contours-%t.ps` for the file name, and the solver saves a file with the appropriate name (e.g., `contours-0001.ps` if the solution is at the first time step).

  This automatic saving of files with the time step should not be used for steady-state cases, since the time step will always remain zero.
- For transient calculations, you can save files with names that reflect the flow-time at which they are saved by including the character string `%f` in the file name. The usage is similar to `%t`. For example, when you specify `filename-%f.ps` for the file name, the solver will save a file with the appropriate name (e.g., `filename-005.000000.ps` for a solution at a flow-time of 5 seconds). By default, the flow-time that is included in the file name will have a field width of 10 and 6 decimal places. To modify this format, use the character string `%`*x* `.` *y* `f`, where *x* and *y* are the preferred field width and number of decimal places, respectively. ANSYS FLUENT will automatically add zeros to the beginning of the flow-time to achieve the prescribed field width. To eliminate these zeros and left align the flow-time, use the character string `%-x .`*y* `f` instead.

  This automatic saving of files with flow-time should not be used for steady-state cases, since the flow-time will always remain zero.
- To save a file with a name that reflects the iteration at which it is saved, use the character string `%I` in the file name. For example, you can specify `contours-%i.ps` for the file name, and the solver saves a file with the appropriate name (e.g., `contours-0010.ps` if the solution is at the 10th iteration).
- To save a picture file with a name that reflects the total number of picture files saved so far in the current solver session, use the character string `%n` in the file name. This option can be used only for picture files.

The default field width for `%I`, `%t`, and `%n` formats is 4. You can change the field width by using `%`*x*`i`, `%`*x*`t`, and `%`*x*`n` in the file name, where *x* is the preferred field width.

4.1.8. Disabling the Overwrite Confirmation Prompt

By default, if you ask ANSYS FLUENT to write a file with the same name as an existing file in that folder, it will ask you to confirm that it is "OK to overwrite" the existing file. If you do not want the solver to ask you for confirmation before it overwrites existing files, you can use the **Batch Options** dialog box (For details, see *Batch Execution Options* (p. 16)). Alternatively, enter the `file/confirm-overwrite?` text command and answer `no` (see *Text User Interface (TUI)* (p. 47) for the text user interface commands).

4.1.9. Toolbar Buttons

The standard toolbar provides buttons that make it easier to read and write files:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Read a file** button ( ) allows you to read existing files using a file selection dialog box. The files available for reading include all those available through the **File/Read** menu item, as described in this chapter.

- The **Write a file** button ( ) allows you to write various types of files. The files available for writing include all those available through the **File/Write** menu item, as described in this chapter.

4.2. Reading Mesh Files

Mesh files are created using the mesh generators (GAMBIT, TGrid, GeoMesh, and PreBFC), or by several third-party CAD packages. From the point of view of ANSYS FLUENT, a mesh file is a subset of a case file (described in *Reading and Writing Case Files* (p. 64)). The mesh file contains the coordinates of all the nodes, connectivity information that tells how the nodes are connected to one another to form faces and cells, and the zone types and numbers of all the faces (e.g., wall-1, pressure-inlet-5, symmetry-2).

The mesh file does not contain any information on boundary conditions, flow parameters. For information about meshes, see *Reading and Manipulating Meshes* (p. 129).

To read a native-format mesh file (i.e., a mesh file that is saved in ANSYS FLUENT format) into the solver, use the **File/Read/Mesh...** menu item. Note that you can also use the **File/Read/Case...** menu item (described in *Reading and Writing Case Files* (p. 64)), because a mesh file is a subset of a case file. GAMBIT, TGrid, GeoMesh, and PreBFC can all write a native-format mesh file. For information about reading these files, see *GAMBIT Mesh Files* (p. 148), *GeoMesh Mesh Files* (p. 149), *TGrid Mesh Files* (p. 149), and *PreBFC Mesh Files* (p. 149).

If after reading in a mesh file (or a case and data file), you would like to read in another mesh file, the *Read Mesh Options Dialog Box* (p. 2047)will open, where you can choose to

- Discard the case and read in a new mesh.
- Replace the existing mesh.

You also have the option to have the **Scale Mesh** dialog box appear automatically for you to check or scale your mesh, which in general is the recommended practice. For this to happen, enable **Show Scale Mesh Panel After Replacing Mesh**.

For information on importing an unpartitioned mesh file into the parallel solver using the partition filter, see *Using the Partition Filter* (p. 1622).

For additional information, please see the following sections:

4.2.1. Reading TGrid Mesh Files
4.2.2. Reading Surface Meshes
4.2.3. Reading GAMBIT and GeoMesh Mesh Files
4.2.4. Reading PreBFC Unstructured Mesh Files

4.2.1. Reading TGrid Mesh Files

TGrid has the same file format as ANSYS FLUENT. Hence you can read a TGrid mesh into the solver using the **File/Read/Mesh...** menu item.

File → **Read** → **Mesh...**

For information about reading TGrid mesh files, see *TGrid Mesh Files* (p. 149).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.2. Reading Surface Meshes

You can read surface mesh files into ANSYS FLUENT using the **Surface Meshes** dialog box, which is opened through the **Geometry Based Adaption** dialog box.

Adapt → **Geometry...**

Click the **Surface Meshes...** button to open the **Surface Meshes** dialog box.

For further details about reading surface mesh files, see *Reading Surface Mesh Files* (p. 160).

4.2.3. Reading GAMBIT and GeoMesh Mesh Files

If you create a FLUENT 5/6, FLUENT/UNS, or RAMPANT mesh in GAMBIT or GeoMesh, you can read it into ANSYS FLUENT using the **File/Read/Mesh...** menu item.

File → **Read** → **Mesh...**

Specify the name of the file to be read the **Select File** dialog box that opens.

4.2.4. Reading PreBFC Unstructured Mesh Files

Since PreBFC's unstructured triangular meshes have the same file format as ANSYS FLUENT, you can read a PreBFC triangular mesh into the solver using the **File/Read/Mesh...** menu item.

File → **Read** → **Mesh...**

Important

Save the file using the `MESH-RAMPANT/TGRID` command.

For information about reading PreBFC mesh files, see *PreBFC Mesh Files* (p. 149).

4.3. Reading and Writing Case and Data Files

Information related to the ANSYS FLUENT simulation is stored in both the case file and the data file. The commands for reading and writing these files are described in the following sections, along with commands for the automatic saving of case and data at specified intervals.

ANSYS FLUENT can read and write either text or binary case and data files. Binary files require less storage space and are faster to read and write. By default, ANSYS FLUENT writes files in binary format. To write a text file, disable the **Write Binary Files** check button in the **Select File** dialog box. In addition, you can read and write either text or binary files in compressed formats (For details, see *Reading and Writing Compressed Files* (p. 59)). ANSYS FLUENT automatically detects the file type when reading.

Furthermore, when writing case and data files, ANSYS FLUENT can perform bzip2 file compression or decompression, which will improve I/O performance by speeding up the overall simulation time. Enable the **Optimize Using Asynchronous I/O** check button in the **Select File** dialog box and enter the file name with a `.bz2` suffix. For example, your file name will be `test.dat.bz2`. ANSYS FLUENT also allows for asynchronous file operations. Note that parallel I/O compression is possible using the **Optimize Using Asynchronous I/O** option.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option is particularly beneficial if you want to improve the overall performance of simulations involving solution check-pointing. Performance is improved by overlapping the file compression and file system copy operations with subsequent iterations or other solver operations. It is best if the front end Cortex process has its own dedicated processor core (or machine) not used by the ANSYS FLUENT compute-node processes, even though the option would aid performance even if they reside on the same machine.

Important

- The **Optimize Using Asynchronous I/O** option is not available on Windows machines.
- If you adapt the mesh, you must save a new case file as well as a data file. Otherwise, the new data file will not correspond to the case file (for example, they will have different numbers of cells). If you have not saved the latest case or data file, ANSYS FLUENT will warn you when you try to exit the program.

For additional information, please see the following sections:

4.3.1. Reading and Writing Case Files
4.3.2. Reading and Writing Data Files
4.3.3. Reading and Writing Case and Data Files Together
4.3.4. Automatic Saving of Case and Data Files

4.3.1. Reading and Writing Case Files

Case files contain the mesh, boundary and cell zone conditions, and solution parameters for a problem. It also contains the information about the user interface and graphics environment. For information about the format of case files see *Appendix B* (p. 2187). The commands used for reading case files can also be used to read native-format mesh files (as described in *Reading Mesh Files* (p. 62)) because the mesh information is a subset of the case information. Select the **File/Read/Case...** menu item to invoke the **Select File** dialog box.

File → **Read** → **Case...**

Read a case file using the **Select File** dialog box. Note that the **Display Mesh After Reading** option in the **Select File** dialog box allows you to have the mesh displayed automatically after it is read.

Select the **File/Write/Case...** menu item to invoke the **Select File** dialog box.

File → **Write** → **Case...**

Write a case file using the **Select File** dialog box.

When ANSYS FLUENT reads a case file, it first looks for a file with the exact name you typed. If a file with that name is not found, it searches for the same file with different extensions (*Reading and Writing Compressed Files* (p. 59)). When ANSYS FLUENT writes a case file, `.cas` is added to the name you type unless the name already ends with `.cas`.

4.3.2. Reading and Writing Data Files

Data files contain the values of the specified flow field quantities in each mesh element and the convergence history (residuals) for that flow field. For information about the format of data files see *Appendix B* (p. 2187).

After reading a mesh or case file, select the **File/Read/Data...** menu item to invoke the **Select File** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → **Read** → **Data...**

Read a data file using the **Select File** dialog box.

After generating data for a case file, select the **File/Write/Data...** menu item to invoke the **Select File** dialog box.

File → **Write** → **Data...**

Write a data file using the **Select File** dialog box.

When ANSYS FLUENT reads a data file, it first looks for a file with the exact name you typed. If a file with that name is not found, it searches for the same file with different extensions (*Reading and Writing Compressed Files* (p. 59)). When ANSYS FLUENT writes a data file, `.dat` is added to the name you type unless the name already ends with `.dat`.

4.3.3. Reading and Writing Case and Data Files Together

A case file and a data file together contain all the information required to restart a solution. Case files contain the mesh, boundary and cell zone conditions, and solution parameters. Data files contain the values of the flow field in each mesh element and the convergence history (residuals) for that flow field.

You can read a case file and a data file together by using the **Select File** dialog box invoked by selecting the **File/Read/Case & Data...** menu item. To read both files, select the appropriate case file, and the corresponding data file (same name with.`dat` suffix) is also read. Note that the **Display Mesh After Reading** option in the **Select File** dialog box allows you to have the mesh displayed automatically after it is read.

File → **Read** → **Case & Data...**

To write a case file and a data file, select the **File/Write/Case & Data...** menu item.

File → **Write** → **Case & Data...**

4.3.4. Automatic Saving of Case and Data Files

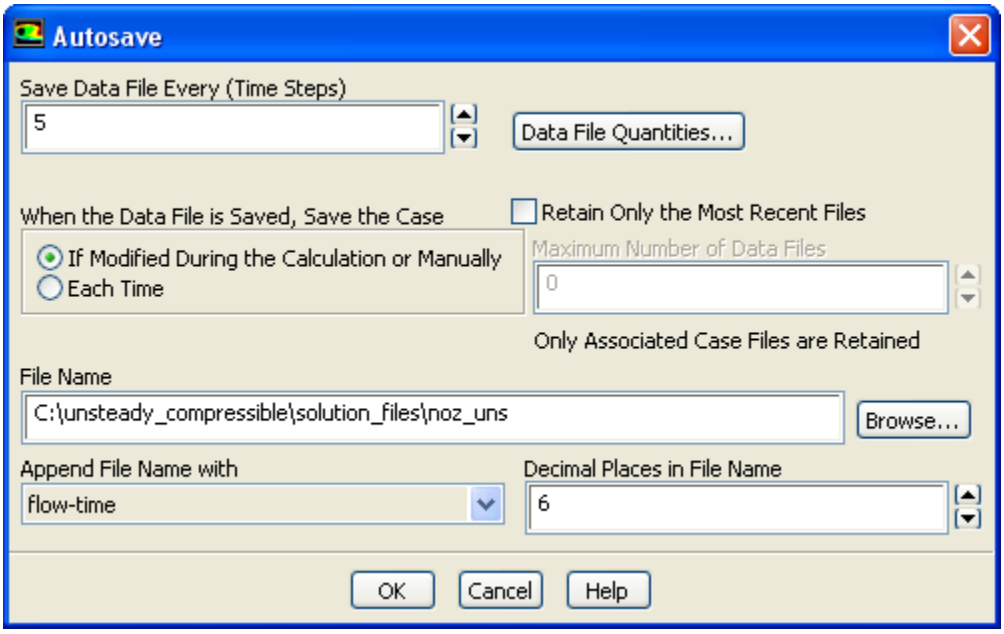
You can request ANSYS FLUENT to automatically save case and data files at specified intervals during a calculation. This is especially useful for time-dependent calculations, since it allows you to save the results at different time steps or flow times without stopping the calculation and performing the save manually. You can also use the autosave feature for steady-state problems, and thus examine the solution at different stages in the iteration history.

Automatic saving is specified using the *Autosave Dialog Box* (p. 1943) (*Figure 4.2* (p. 66)), which is opened by clicking the **Edit...** button next to the **Autosave Every** text box in the **Calculation Activities** task page.

Calculation Activities

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.2 The Autosave Dialog Box



Specify how often you would like to save your modified files by entering the frequency in the **Save Data File Every** number-entry field. **Save Data File Every** is set to zero by default, indicating that no automatic saving is performed.

If you choose to save the case file only if it is modified, then select **If Modified During the Calculation or Manually** under **When the Data File is Saved, Save the Case File**. Note that the case file will be saved whether you make a manual change, or if ANSYS FLUENT makes a change internally during the calculation. If you choose to save the case file every time the data file is saved, then select **Each Time**.

Important

To save only the data files, use the following TUI option:

`file` → `auto-save` → `case-frequency` → `if-mesh-is-modified`

This will result in the options in the **When the Data File is Saved, Save the Case** group box being disabled in the **Autosave** dialog box. In essence, this TUI command forces ANSYS FLUENT to the save case file only when the mesh is modified. It does not disable case file saving, but reduces it to an absolute minimum. This is necessary to do so since you cannot read a data file without a case file containing a matching mesh.

For steady-state solutions, specify the frequency in iterations. For transient solutions, specify it in time steps (unless you are using the explicit time stepping formulation, in which case specify the frequency in iterations). If you define a frequency of 5, for example, a case file is saved every 5 iterations or time steps.

If you have limited disk space, restrict the number of files saved by ANSYS FLUENT by enabling the **Retain Only the Most Recent Files** option. When enabled, enter the **Maximum Number of Data Files** you would like to retain. Note that the case and data files are treated separately with regard to the maximum number of files saved when overwriting. For example, if the value of **Maximum Number of Data Files** is set to five, ANSYS FLUENT saves a maximum of five case and five data files, irrespective of the frequency. After the maximum limit of files has been saved, ANSYS FLUENT begins overwriting the earliest existing file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have generated data (either by initializing the solution or running the calculation) you can view the list of standard quantities that will be written to the data file as a result of the autosave, and even select additional quantities for postprocessing in alternative applications. Click the **Data File Quantities...** button to open the **Data File Quantities** dialog box, and make any necessary selections. For details, see *Setting Data File Quantities* (p. 121).

Enter a root name for the autosave files in the **File Name** text box. When the files are saved, a number will be appended to this root name to indicate the point at which it was saved during the calculation: for steady-state solutions, this will be the iteration number, whereas for transient solutions it will be either the time step number or flow time (depending on your selection in the step that follows). An extension will also be automatically added to the root name (`.cas` or `.dat`). If the specified **File Name** ends in `.gz` or `.Z`, appropriate file compression is performed. For details about file compression see *Reading and Writing Compressed Files* (p. 59).

For transient calculations, make a selection from the **Append File Name with** drop-down list to indicate whether you want the root file name to be appended with the **time-step** or **flow-time** (see *Figure 4.2* (p. 66)). If you select the latter, you can set the **Decimal Places in File Name** to determine the ultimate width of the file name.

Consider a transient case for which you want to save your case and data files at known time steps. The procedure you would follow is to first set the frequency in the **Save Data File Every** text box. Select **Each Time** if you want both case and data files saved at the same interval. Then enter `my_file` for the **File Name**. Finally, select **time-step** from the **Append File Name with** drop-down list. An example of the resulting files saved would be

```
my_file-0005.cas
```

```
my_file-0005.dat
```

indicating that these files were saved at the fifth time step.

You can revise the instructions for the previous example to instead save case and data files at known flow times, by selecting **flow-time** from the **Append File Name with** drop-down list. The default **Decimal Places in File Name** will be six. An example of the resulting files saved would be

```
my_file-0.500000.cas
```

```
my_file-0.500000.dat
```

indicating that these files were saved at a flow time of 0.5 seconds.

For steady-state and transient cases, you have the option of automatically numbering the files (as described in *Automatic Numbering of Files* (p. 61)), and thereby include further information about when the files were saved. This involves the addition of special characters to the **File Name**. For example, you may want the file names to convey the flow times with their corresponding time steps (transient cases only). Select **time-step** from the **Append File Name with**, and enter a **File Name** that ends with `-%f` to automatically number the files with the flow time. Thus, entering a **File Name** of `filename-%f` could result in a saved case file named `filename-000.500000-0010.cas`. The conventions used in this example can be explained as follows:

- `filename-` is the file name you entered when autosaving your solution.
- `000.500000` is the result of the special character `%f` added to the file name, and is the flow time. This flow time has a field width of ten characters, which allows for six decimal places (as discussed in *Automatic Numbering of Files* (p. 61)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `-0010` is the appended **time-step**, as designated by the selection in the **Append File Name with** drop-down list.
- `.cas` is the file extension automatically added when using the autosave option.

All of the autosave inputs are stored in memory when you click **OK** in the **Autosave** dialog box, and can then be saved with the case file.

Important

The **Autosave** dialog box is slightly different when running ANSYS FLUENT within ANSYS Workbench. For more information, see the ANSYS FLUENT in Workbench User's Guide.

4.4. Reading and Writing Parallel Data Files

When performing parallel processing, you have the option of utilizing the parallel input/output (I/O) capability of ANSYS FLUENT. This capability permits the writing and reading of data files directly to and from the node processes in a parallel fashion. This is in contrast to the standard ANSYS FLUENT I/O approach, in which all data is passed from the node processes to the host process where it is then written out to disk in serial. The motivation for performing the I/O directly from the node processes is to reduce the time for the data file I/O operations.

During parallel I/O operations, all the node processes write to or read from the same file, that is, they concurrently access a single file. The format of this parallel data file is different than the format of the standard ANSYS FLUENT data file. The format has been revised to permit more efficient concurrent access. The order of data written to the parallel data file is dependent on the number of processes involved in the session and on the partitioning. However, a parallel data file written from any parallel session on a given platform (i.e., any number of processes or partitioning) can be read by any other parallel session on the same platform. In other words, analyses may be freely restarted using a different number of processes, as long as the files are being written or read from the same platform.

For information about parallel processing, see *Parallel Processing* (p. 1585).

For additional information, please see the following sections:

4.4.1. Writing Parallel Data Files
4.4.2. Reading Parallel Data Files
4.4.3. Availability and Limitations

4.4.1. Writing Parallel Data Files

To write a parallel data file, select the **File/Write/Data...** menu item to invoke the **Select File** dialog box.

File → **Write** → **Data...**

In the **Select File** dialog box, enter the name of the output data file with a `.pdat` extension. The parallel data files are always binary, so you do not have to enable the **Write Binary Files** option.

Note that parallel data files written from sessions with different numbers of processes may be slightly different in size. However, they contain essentially the same information.

You can also use the text command `file/write-pdat?` to write a `.pdat` file. This is particularly useful if you want to write a `.pdat` file using the autosave data file option, after reading in a data file. The `.pdat` files will be saved during the autosave.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.2. Reading Parallel Data Files

To read parallel data files, select the **File/Read/Data...** menu item to invoke the **Select File** dialog box.

File → **Read** → **Data...**

In the **Select File** dialog box, enter the name of the input data file with a `.pdat` extension.

Note that a parallel file will be read slightly quicker into a session that uses the same number of processes as the session that originally wrote the file, as opposed to a session with a different number.

4.4.3. Availability and Limitations

The conditions in which parallel I/O is available and the limitations related to this capability are as follows:

- The parallel I/O capability is only available for parallel ANSYS FLUENT sessions. Parallel data files cannot be read into a serial ANSYS FLUENT session, although they can be read if ANSYS FLUENT is launched with the number of processes set to one (e.g., enter `1` for **Number of Processes** in FLUENT Launcher).
- On a given platform, compatibility of data files will be a function of MPI type, but not interconnect.
  - Within the same MPI on a given platform, data files will be compatible regardless of the interconnect (i.e., writing with HP-MPI/ `ethernet` and reading with HP-MPI/ `infiniband` is allowed).
  - Inter-MPI compatibility on the same platform may also exist, since most of the MPIs use the same MPI-O implementation (ROMIO), but this is not guaranteed.
- Parallel data files written on one platform may not necessarily be able readable on other platforms. For example, compatibility across 64-bit platforms with the same endianness (byte order) may even exist (e.g., `lnia64` and `lnamd64`); however, this is not guaranteed by the standard (since the "native" format is used) and should be investigated on a case-by-case basis.
- No parallel I/O capabilities are currently available for case files.
- The parallel/IO capability is designed to work on true parallel file systems designed for concurrent, single-file access. The capability is not supported on NFS, for example.
- File compression with parallel I/O is supported via asynchronous operation. Inline compression as with standard I/O is not possible. (For details, see *Reading and Writing Case and Data Files* (p. 63)).
- The general parallel I/O capability is only available with certain MPIs on certain platforms. The supported combinations are listed in *Table 4.1: Architecture/MPI Combinations Compatible with Parallel I/O* (p. 69). Some of the MPIs currently used on Linux/UNIX systems do not support MPI-2 features used in the parallel I/O. The default MPIs on all Linux and Windows systems should support the capability.

Important

Not all MPIs on a given platform work with all interconnects. For information about compatibility, see *Table 34.6: Supported MPIs for Linux/UNIX Architectures (Per Interconnect)* (p. 1601).

Table 4.1 Architecture/MPI Combinations Compatible with Parallel I/O

Architecture	MPI
`lnx86`	`hp`, `mpich2`
`lnamd64`	`hp`, `intel`, `openmpi`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Architecture	MPI
lnia64	hp, intel, sgi
ntx86	hp, intel
win64	hp, intel, ms
aix51_64	openmpi, vendor
hpux11_ia64	vendor
ultra_64	vendor

- The parallel I/O capability is available with certain file systems on certain platforms. The supported combinations are listed in *Table 4.2: Architecture/File System with Parallel I/O* (p. 70).

Table 4.2 Architecture/File System with Parallel I/O

Architecture	File System
aix51_64	GPFS
lnamd64	Panasas, GPFS, SFS, LUSTRE, PVFS2, MPFS

4.5. Reading FLUENT/UNS and RAMPANT Case and Data Files

Case files created by FLUENT/UNS 3 or 4 or RAMPANT 2, 3, or 4 can be read into ANSYS FLUENT in the same way that current case files are read (see *Reading and Writing Case and Data Files* (p. 63)). If you read a case file created by FLUENT/UNS, ANSYS FLUENT selects **Pressure-Based** in the **Solver** group box of the *General Task Page* (p. 1631). If you read a case file created by RAMPANT, ANSYS FLUENT selects **Density-Based** in the **Solver** group box of the *General Task Page* (p. 1631), as well as **Explicit** from the **Formulation** drop-down menu in the *Solution Methods Task Page* (p. 1900).

Data files created by FLUENT/UNS 4 or RAMPANT 4 can be read into ANSYS FLUENT in the same way that current data files are read (see *Reading and Writing Case and Data Files* (p. 63)).

4.6. Reading and Writing Profile Files

Boundary profiles are used to specify flow conditions on a boundary zone of the solution domain. For example, they can be used to prescribe a velocity field on an inlet plane. For information on boundary profiles, see *Profiles* (p. 366). For information about transient profiles, see *Defining Transient Cell Zone and Boundary Conditions* (p. 376).

For additional information, please see the following sections:

4.6.1. Reading Profile Files
4.6.2. Writing Profile Files

4.6.1. Reading Profile Files

To read the boundary profile files, invoke the **Select File** dialog box by selecting the **File/Read/Profile...** menu item.

File → **Read** → **Profile...**

This opens the **Select File** dialog box so that you can read a boundary profile with the standard extension `.prof` or a transient profile in tabular format with the standard extension `.ttab`. If a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

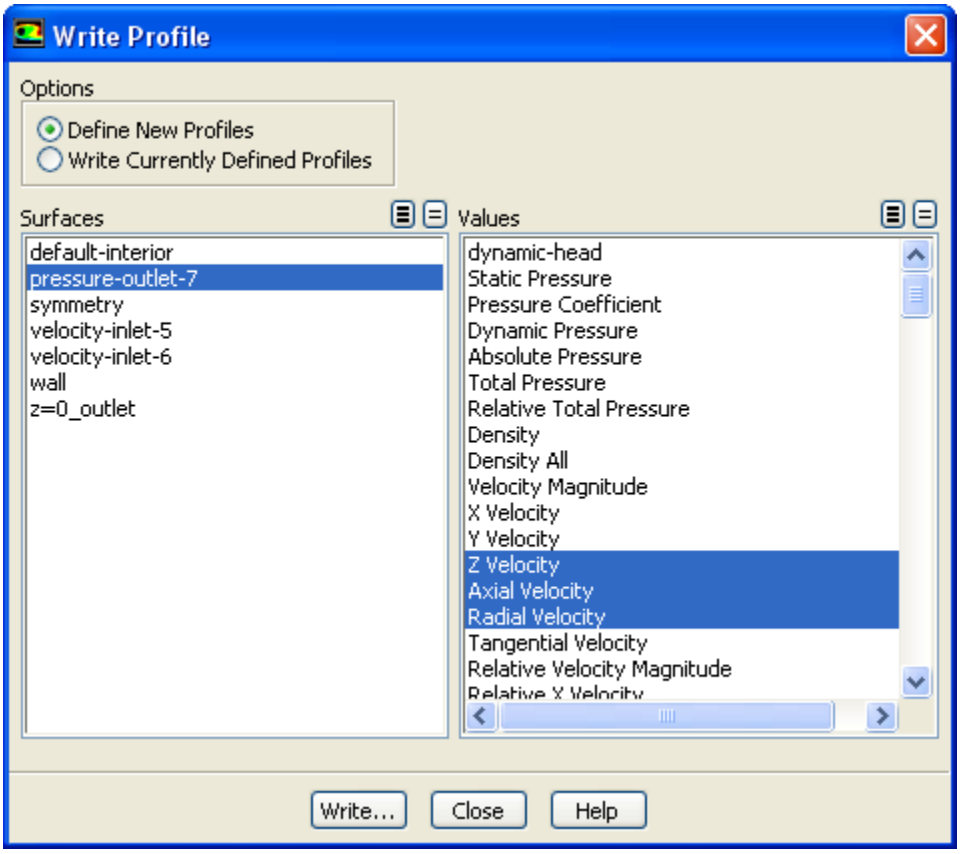
4.6.2. Writing Profile Files

You can also create a profile file from the conditions on a specified boundary or surface. For example, you can create a profile file from the outlet conditions of one case. Then you can read that profile into another case and use the outlet profile data as the inlet conditions for the new case.

To write a profile file, use the *Write Profile Dialog Box* (p. 1811) (*Figure 4.3* (p. 71)).

File → **Write** → **Profile...**

Figure 4.3 The Write Profile Dialog Box



1. Retain the default option of **Define New Profiles**.
2. Select the surface(s) from which you want to extract data for the profile(s) in the **Surfaces** list.
3. Choose the variable(s) for which you want to create profiles in the **Values** list.
4. Click **Write...** and specify the profile file name in the resulting **Select File** dialog box.

   ANSYS FLUENT saves the mesh coordinates of the data points for the selected surface(s) and the values of the selected variables at those positions. When you read the profile file back into the solver, you can select the profile values from the relevant drop-down lists in the boundary condition dialog boxes. The names of the profile values in the drop-down lists will consist of the surface name and the particular variable.
5. Select the **Write Currently Defined Profiles** option:
   - if you have made any modifications to the boundary profiles since you read them in (e.g., if you reoriented an existing profile to create a new one).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- if you wish to store all the profiles used in a case file in a separate file.

6. Click **Write...** and specify the file name in the resulting **Select File** dialog box. All currently defined profiles are saved in this file. This file can be read back into the solver whenever you wish to use these profiles again.

4.7. Reading and Writing Boundary Conditions

To save all currently defined boundary conditions to a file, enter the `file/write-settings` text command and specify a name for the file.

`file` → `write-settings`

ANSYS FLUENT writes the boundary and cell zone conditions, the solver, and model settings to a file using the same format as the "zone" section of the case file. See *Appendix B* (p. 2187) for details about the case file format.

To read boundary conditions from a file and to apply them to the corresponding zones in your model, enter the `file/read-settings` text command.

`file` → `read-settings`

ANSYS FLUENT sets the boundary and cell zone conditions in the current model by comparing the zone name associated with each set of conditions in the file with the zone names in the model. If the model does not contain a matching zone name for a set of boundary conditions, those conditions are ignored.

If you read boundary conditions into a model that contains a different mesh topology (e.g., a cell zone has been removed), check the conditions at boundaries within and adjacent to the region of the topological change. This is important for wall zones.

Note

If the boundary conditions are not checked and some remain uninitialized, the case will not run successfully.

When the `file/read-settings` text command is not used, all boundary conditions get the default settings when a mesh file is imported, allowing the case to run with the default values.

If you want ANSYS FLUENT to apply a set of conditions to multiple zones with similar names, or to a single zone with a name you are not sure of in advance, you can edit the boundary-condition file saved with the `file/write-settings` command to include *wildcards* (`*`) within the zone names. For example, if you want to apply a particular set of conditions to **wall-12**, **wall-15**, and **wall-17** in your current model, edit the boundary-condition file so that the zone name associated with the desired conditions is `wall-*`.

4.8. Writing a Boundary Mesh

You can write the boundary zones (surface mesh) to a file. This file can be read and used by TGrid to produce a volume mesh. You may find this feature useful if you are unsatisfied with a mesh obtained from another mesh generation program.

A boundary mesh can be written using the **Select File** dialog box invoked by selecting the **File/Write/Boundary Mesh...** menu item.

File → **Write** → **Boundary Mesh...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.9. Reading Scheme Source Files

A Scheme source file can be loaded in three ways: through the menu system as a scheme file, through the menu system as a journal file, or through Scheme itself.

For large source files, use the **Select File** dialog box invoked by selecting the **File/Read/Scheme...** menu item

File → **Read** → **Scheme...**

or use the Scheme `load` function in the console, as shown in the following example:

```
> (load "file.scm")
```

Shorter files can also be loaded with the **File/Read/Journal...** menu item or the `file/ read-journal` command in the text interface (or its `.` or `source` alias, as shown in the example that follows).

```
>. file.scm

> source file.scm
```

In this case, each character of the file is echoed to the console as it is read, in the same way as if you were typing in the contents of the file.

4.10. Creating and Reading Journal Files

A journal file contains a sequence of ANSYS FLUENT commands, arranged as they would be typed interactively into the program or entered through the GUI or TUI. The GUI and TUI commands are recorded as Scheme code lines in journal files. You can also create journal files manually with a text editor. If you want to include comments in your file, be sure to put a semicolon (`;`) at the beginning of each comment line. See *Background Execution on Linux/UNIX Systems* (p. 14) for an example.

The purpose of a journal file is to automate a series of commands instead of entering them repeatedly on the command line. Another use is to produce a record of the input to a program session for later reference, although transcript files are often more useful for this purpose. (For details, see *Creating Transcript Files* (p. 75)).

Command input is taken from the specified journal file until its end is reached, at which time control is returned to the standard input (usually the keyboard). Each line from the journal file is echoed to the standard output (usually the screen) as it is read and processed.

Important

A journal file is, by design, just a simple record and playback facility. It contains no information about the state in which it was recorded or the state in which it is being played back.

- Be careful not to change the folder while recording a journal file. Also, try to re-create the state in which the journal was written before you read it into the program. For example, if your journal file includes an instruction to save a new file with a specified name, you should check that if a file with that name exists in your folder before you read in your journal file. If a file with that name exists and you read in your journal file, when the program reaches the write instruction, it will prompt for a confirmation if it is OK to overwrite the old file.

  Since the journal file does not contain any response to the confirmation request, ANSYS FLUENT cannot continue to follow the instructions of the journal file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Other conditions that may affect the program's ability to perform the instructions contained in a journal file can be created by modifications or manipulations that you make within the program.

  For example, if your journal file creates several surfaces and displays data on those surfaces, you must be sure to read in appropriate case and data files before reading the journal file.

Important

At a point of time, only one journal file can be open for recording, but you can write a journal and a transcript file simultaneously. You can also read a journal file at any time.

Whether you choose to type the text command in full or use partial strings (as described in *Command Abbreviation* (p. 48)), complete commands are recorded in the journal files. Consider the following examples:

- Typing in the TUI

  ```
  solve/set/expert , , yes , ,
  ```

  will be recorded in the journal file as

  ```
  /solve/set/expert yes no yes no no
  ```

  where `,` or <Enter> signifies default values or entries, as described in *Text Prompt System* (p. 50).

- Typing in the TUI

  ```
  so set ur mom 0.2 pres 0.4
  ```

  will be recorded in the journal file as two separate commands:

  ```
  /solve/set/under-relaxation/mom 0.2
  /solve/set/under-relaxation/pressure 0.4
  ```

Important

- Only successfully completed commands are recorded. For example, if you stopped an execution of a command using the `<Ctrl>` key and the `c` key, it will not be recorded in the journal file.
- If a GUI event happens while a text command is in progress, the GUI event is recorded first.
- All default values are recorded (as in the first example above).

For additional information, please see the following section:
4.10.1. Procedure

4.10.1. Procedure

To start the journaling process, select the **File/Write/Start Journal...** menu item.

File → **Write** → **Start Journal...**

After you enter a name for the file in the **Select File** dialog box, journal recording begins. The **Start Journal...** menu item becomes the **Stop Journal** menu item. You can end journal recording by selecting **Stop Journal**, or by exiting the program.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → **Write** → **Stop Journal**

You can read a journal file into the program using the **Select File** dialog box invoked by selecting the **File/Read/Journal...** menu item.

File → **Read** → **Journal...**

Journal files are always loaded in the main (i.e., top-level) text menu, regardless of where you are in the text menu hierarchy when you invoke the read command.

4.11. Creating Transcript Files

A transcript file contains a complete record of all standard input to and output from ANSYS FLUENT (usually all keyboard and GUI input and all screen output). GUI commands are recorded as Scheme code lines in transcript files. ANSYS FLUENT creates a transcript file by recording everything typed as input or entered through the GUI, and everything printed as output in the text window.

The purpose of a transcript file is to produce a record of the program session for later reference. Because they contain messages and other output, transcript files (unlike journal files), cannot be read back into the program.

Important

Only one transcript file can be open for recording at a time, but you can write a transcript and a journal file simultaneously. You can also read a journal file while a transcript recording is in progress.

To start the transcription process, select the **File/Write/Start Transcript...** menu item.

File → **Write** → **Start Transcript...**

After you enter a name for the file in the **Select File** dialog box, transcript recording begins and the **Start Transcript...** menu item becomes the **Stop Transcript** menu item.

You can end transcript recording by selecting **Stop Transcript**, or by exiting the program.

File → **Write** → **Stop Transcript**

4.12. Importing Files

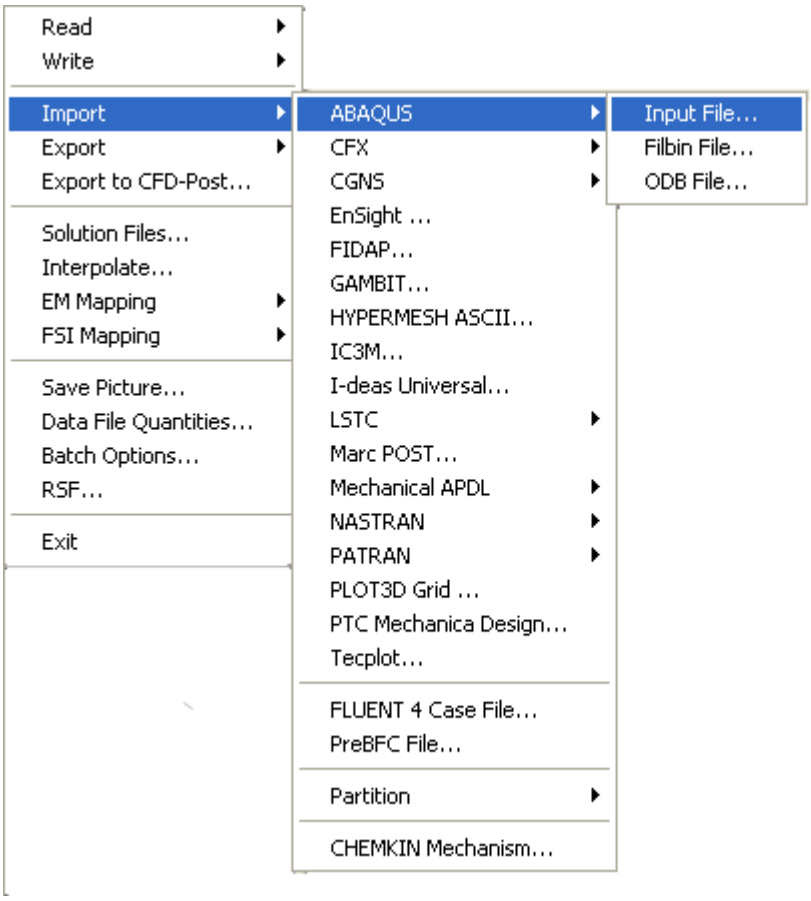
ANSYS FLUENT allows you to import the following file formats:

- ABAQUS `.inp`, `.fil`, and `.odb` files.
- ANSYS CFX `.def` and `.res` files.
- CGNS files.
- EnSight files.
- ANSYS FIDAP files.
- GAMBIT files.
- HYPERMESH ASCII files.
- IC3M files.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- I-deas Universal files.
- LSTC/DYNA keyword input files and state databases.
- Marc POST files.
- Mechanical APDL `.inp`, `.cdb`, `.rst`, `.rmg`, and `.rfl` files.
- NASTRAN Bulk Data files.
- PATRAN Neutral files.
- PLOT3D mesh files.
- PTC Mechanica Design studies.
- Tecplot files.

Figure 4.4 The Import Menu



For information on importing particle history data, see *Importing Particle Data* (p. 1070).

For additional information, please see the following sections:

4.12.1. ABAQUS Files
4.12.2. ANSYS CFX Files
4.12.3. Meshes and Data in CGNS Format
4.12.4. EnSight Files
4.12.5. ANSYS FIDAP Neutral Files
4.12.6. GAMBIT and GeoMesh Mesh Files
4.12.7. HYPERMESH ASCII Files
4.12.8. IC3M Files

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.12.9. I-deas Universal Files
4.12.10. LSTC Files
4.12.11. Marc POST Files
4.12.12. Mechanical APDL Files
4.12.13. NASTRAN Files
4.12.14. PATRAN Neutral Files
4.12.15. PLOT3D Files
4.12.16. PTC Mechanica Design Files
4.12.17. Tecplot Files
4.12.18. FLUENT 4 Case Files
4.12.19. PreBFC Files
4.12.20. Partition Files
4.12.21. CHEMKIN Mechanism

4.12.1. ABAQUS Files

To import an ABAQUS input file, use the **File/Import/ABAQUS/Input File...** menu item.

File → **Import** → **ABAQUS** → **Input File...**

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **Input File** to be read. The ABAQUS input file (.inp) is a text file which contains the input description of a finite element model for the ABAQUS finite element program. The interface only produces datasets associated with the finite element model, no results of datasets are produced. Element types commonly associated with structural analysis are supported by this file format. There is a list of input keywords that are recognized in the ABAQUS **Input File**[*3*] (p. 2209).

To import an ABAQUS filbin file, use the **File/Import/ABAQUS/Filbin File...** menu item.

File → **Import** → **ABAQUS** → **Filbin File...**

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **Filbin File** to be read. This output file has a `.fil` extension and consists of finite element model and results data.

To import an ABAQUS ODB file, use the **File/Import/ABAQUS/ODB File...** menu item.

File → **Import** → **ABAQUS** → **ODB File...**

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **ODB File** to be read. This output database file has a `.odb` extension and consists of finite element model and results data in the OpenDocument format.

Important

Importing ABAQUS ODB files is available on a limited number of platforms. For more information refer to the ANSYS FLUENT 13 release notes.

4.12.2. ANSYS CFX Files

To import an ANSYS CFX definition file, use the **File/Import/CFX/Definition File...** menu item.

File → **Import** → **CFX** → **Definition File...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select this menu item to invoke the **Select File** dialog box. Specify the name of the ANSYS CFX **Definition File** to be read. The solver reads mesh information from the ANSYS CFX file with `.def` extensions. For information about importing ANSYS CFX files, see *ANSYS CFX Files* (p. 154).

To import an ANSYS CFX result file, use the **File/Import/CFX/Result File...** menu item.

File → **Import** → **CFX** → **Result File...**

In the **Select File** dialog box, specify the name of the ANSYS CFX **Result File** to be read. Those imported files will have `.res` extensions.

Note that the **Create Zones from CCL Physics Data** option in the **Select File** dialog boxes allows you to create zones from the physics data objects or the primitive mesh region objects.

Important

ANSYS CFX file import is available for 3D cases only.

4.12.3. Meshes and Data in CGNS Format

To import meshes in CFD general notation system (CGNS) format (`.cgns`) into ANSYS FLUENT, use the **File/Import/CGNS/Mesh...** menu item.

File → **Import** → **CGNS** → **Mesh...**

To import a mesh and the corresponding CGNS data, use the **File/Import/CGNS/Mesh & Data...** menu item.

File → **Import** → **CGNS** → **Mesh & Data...**

To import only the CGNS data, use the **File/Import/CGNS/Data...** menu item.

File → **Import** → **CGNS** → **Data...**

Important

- To import data correctly, first import the mesh using the mesh only option (**Mesh...**), set up the boundary conditions, and read the data using the data only option (**Data...**). For example, if a boundary zone is of type **pressure-outlet** and is read as **outlet**, it should be changed to **pressure-outlet** before importing the data.
- The new and original meshes should have the same zones, numbered in the same order. A warning is issued if they do not, because inconsistencies can create problems with the boundary conditions.

4.12.4. EnSight Files

You can import an EnSight file using the **File/Import/EnSight...** menu item.

File → **Import** → **EnSight...**

This file format is applied to both unstructured and structured data, where each part contains its own local coordinate array. The EnSight Gold software package, which uses this file format, allows you to analyze,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

visualize, and communicate engineering datasets. It allows the you to take full advantage of parallel processing and rendering and supports a range of virtual reality devices. Furthermore, it enables real-time collaboration.

When selecting this option, the **Select File** dialog box will appear, where you will specify a file name. This file will have an `.encas` or `.case` extension.

Only the mesh file is read into ANSYS FLUENT, and any data present is discarded.

4.12.5. ANSYS FIDAP Neutral Files

You can read a menu item.

File → **Import** → **FIDAP...**

In the **Select File** dialog box, specify the name of the ANSYS FIDAP **Neutral File** to be read. This file will have an `.FDNEUT` or `.unv` file extension. ANSYS FLUENT reads mesh information and zone types from the ANSYS FIDAP file. You must specify boundary conditions and other information after reading this file. For information about importing ANSYS FIDAP Neutral files, see *ANSYS FIDAP Neutral Files* (p. 157).

4.12.6. GAMBIT and GeoMesh Mesh Files

If you have saved a neutral file from GAMBIT, rather than an ANSYS FLUENT mesh file, you can import it into ANSYS FLUENT using the **File/Import/GAMBIT...** menu item.

File → **Import** → **GAMBIT...**

For information about importing files from GAMBIT and GeoMesh, see *GAMBIT Mesh Files* (p. 148) and *GeoMesh Mesh Files* (p. 149).

4.12.7. HYPERMESH ASCII Files

You can read a HYPERMESH ASCII file using the **File/Import/HYPERMESH ASCII...** menu item.

File → **Import** → **HYPERMESH ASCII...**

HYPERMESH is a high-performance finite element pre- and postprocessor for popular finite element solvers, allowing engineers to analyze product design performance in a highly interactive and visual environment.

When selecting this option, the **Select File** dialog box will appear, where you will specify a file name. This file should have an `.hm`, `.hma`, or `.hmascii` extension.

4.12.8. IC3M Files

ANSYS FLUENT has the ability to import in-cylinder simulation files that have been created using the IC3M preprocessor of ICEM CFD. For information about creating and setting up such files, see the IC3M User Guide and Tutorial Manual (available to licensed ICEM CFD IC3M users).

When importing IC3M files, first make sure that all of the relevant files exported from ICEM CFD IC3M are in the same folder and uncompressed. Then import the files, using the **File/Import/IC3M...** menu item.

File → **Import** → **IC3M...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- IC3M files can only be imported into the 3D version of ANSYS FLUENT (single or double precision).
- IC3M files should not be used with ANSYS FLUENT in Workbench.

In the **Select File** dialog box, specify the name of the mesh file, which should have a `.msh` extension. After the file is selected, ANSYS FLUENT will read in the mesh and automatically set up the case file to run a dynamic mesh simulation, using the parameters defined in the IC3M preprocessor. You can make certain changes to the dynamic mesh setup, but this is not necessary. See *Using Dynamic Meshes* (p. 548) for further details about setting up the dynamic mesh. Note that you cannot change the engine parameters for an engine set up using IC3M, except for the time step size.

The information read in from the IC3M files includes the starting and ending crank angle for the simulation. After reading the `.msh` file, ANSYS FLUENT will position the mesh such that it corresponds to the starting crank angle. This process may take a few minutes. You will be informed of the status of this process through messages in the console.

Save the case file at this point, to retain a copy of the settings from the IC3M files. You may start the simulation from this case file, as long as it is in a folder that contains *all* of the uncompressed files exported by ICEM CFD IC3M. It is recommended that you preview the mesh motion prior to running the simulation, using the same time step size that will be used in the simulation. See *Previewing the Dynamic Mesh* (p. 626) for further details.

Important

- You will not be able to run the mesh preview or simulation beyond the ending crank angle specified in the IC3M preprocessor, even if the motion is periodic. ANSYS FLUENT sets the end crank angle based on the information imported from the IC3M files, and will automatically stop when this angle is reached.
- You cannot reuse the same ANSYS FLUENT session for engines set up using IC3M, i.e., for each new calculation, you must exit ANSYS FLUENT and begin a new session.

4.12.9. I-deas Universal Files

I-deas Universal files can be read into ANSYS FLUENT with the **File/Import/I-deas Universal...** menu item.

File → **Import** → **I-deas Universal...**

Select the **I-deas Universal...** menu item to invoke the **Select File** dialog box.

Specify the name of the I-deas Universal file to be read. The solver reads mesh information and zone types from the I-deas Universal file. For information about importing I-deas Universal files, see *I-deas Universal Files* (p. 150).

4.12.10. LSTC Files

To import an LSTC input file, use the **File/Import/LSTC/Input File...** menu item.

File → **Import** → **LSTC** → **Input File...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The LSTC input file is a text file which contains the input description of a finite element model for the LS-DYNA finite element program. This interface only produces datasets associated with the mesh, no results datasets are produced. The element types commonly associated with structural analysis are supported.

LSTC input files have the following file extensions: `.k`, `.key`, and `.dyn`

To import an LSTC state file, use the **File/Import/LSTC/State File...** menu item.

File → **Import** → **LSTC** → **State File...**

The state file consists of three major sections: control data, geometry data, and state data. Each dataset in the state data section corresponds to the time and global data items associated with each state on the database. Dataset attributes include such things as time, energy, and momentum.

An LSTC state file has a `.d3plot` file extension.

4.12.11. Marc POST Files

Marc POST files can be read into ANSYS FLUENT using the **File/Import/Marc POST...** menu item.

File → **Import** → **Marc POST...**

Select the **Marc POST...** menu item and in the **Select File** dialog box, specify the name of the file to be read.

These files are generated using MSC Marc, a nonlinear finite element program. MSC Marc allows you to study deformations that exceed the linear elastic range of some materials, enabling you to assess the structural integrity and performance of the material. It also allows you to simulate deformations that are part-to-part or part-to-self contact under a range of conditions.

4.12.12. Mechanical APDL Files

To import a Mechanical APDL input file, use the **File/Import/Mechanical APDL/Input File...** menu item.

File → **Import** → **Mechanical APDL** → **Input File...**

Select this menu item to invoke the **Select File** dialog box. Specify the name of the Mechanical APDL **Prep7 File** to be read. The solver reads mesh information from the Mechanical APDL file with `.ans`, `.neu`, `.cdb`, and `.prep7` extensions. For information about importing Mechanical APDL files, see *Mechanical APDL Files* (p. 153).

To import a Mechanical APDL result file, use the **File/Import/Mechanical APDL/Result File...** menu item.

File → **Import** → **Mechanical APDL** → **Result File...**

In the **Select File** dialog box. Specify the name of the Mechanical APDL **Result File** to be read. Those imported files will have `.rfl`, `.rst`, `.rth`, and `.rmg` extensions.

4.12.13. NASTRAN Files

You can read NASTRAN Bulkdata files into ANSYS FLUENT with the **File/Import/NASTRAN/Bulkdata File...** menu item.

File → **Import** → **NASTRAN** → **Bulkdata File...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select the **Bulkdata File...** menu item, the **Select File** dialog box will appear and you will specify the name of the NASTRAN **File** to be read. This file will have `.nas`, `.dat`, `.bdf` file extensions. The solver reads mesh information from the NASTRAN file. For information about importing NASTRAN files, see *NASTRAN Files* (p. 151).

To import NASTRAN Op2 files into ANSYS FLUENT, use the **File/Import/NASTRAN/Op2 File...** menu item.

File → **Import** → **NASTRAN** → **Op2 File...**

In the **Select File** dialog box, specify the name of the NASTRAN **Output2 File** to be read. This file is an output binary data file that contains data used in the NASTRAN finite element program. This file will have `.op2` file extension.

4.12.14. PATRAN Neutral Files

To read a PATRAN Neutral file zoned by named components (that is, a file in which you have grouped nodes with the same specified group name), use the **File/Import/PATRAN/Neutral File...** menu item.

File → **Import** → **PATRAN** → **Neutral File...**

Selecting this menu item invokes the **Select File** dialog box. Specify the name of the `.neu`, `.out`, or `.pat` PATRAN Neutral file to be read. The solver reads mesh information from the PATRAN Neutral file. For information about importing PATRAN Neutral files, see *PATRAN Neutral Files* (p. 152).

To read a PATRAN **Result File**, use the **File/Import/PATRAN/Result File...** menu item.

File → **Import** → **PATRAN** → **Result File...**

In the **Select File** dialog box, specify the name of the PATRAN **Result File** to be read.

4.12.15. PLOT3D Files

To import a PLOT3D mesh file, use the **File/Import/PLOT3D Grid...** menu item.

File → **Import** → **PLOT3D Grid...**

The PLOT3D mesh files have `.p3d`, `.bin`, `.x`, `.xyz`, or `.grd` file extensions.

These file formats may be formatted, unformatted or binary.

4.12.16. PTC Mechanica Design Files

To import a PTC Mechanica Design file, use the **File/Import/PTC Mechanica Design...** menu item.

File → **Import** → **PTC Mechanica Design...**

This will open the **Select File** dialog box. Specify the name of the neutral file to be read.

The PTC Mechanica Design file contains analysis, model and results data. Only the binary form of the results data files is supported.

The form of the file must have the `.neu` extension.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Mechanica results consists of an entire directory structure of files, called a "study" in Mechanica terminology, which must be used in exactly the form that Mechanica originally generates it. ANSYS FLUENT's VKI interface keys on the `.neu` file and can traverse the directory structure from there to access the other files that it needs.

4.12.17. Tecplot Files

To import a Tecplot file, use the **File/Import/Tecplot...** menu item.

File → **Import** → **Tecplot...**

This will open the **Select File** dialog box. Specify the name of the neutral file to be read.

The Tecplot file is a binary file. Only the mesh is read into ANSYS FLUENT and any data present is discarded.

The form of the file must have the `.plt` extension. ANSYS FLUENT supports the importation of polyhedral cells and files created by Tecplot version 7.1–11.2, except for version 11.0 (which is unsupported).

4.12.18. FLUENT 4 Case Files

You can read a FLUENT 4 case file using the **File/Import/FLUENT 4 Case File...** menu item.

File → **Import** → **FLUENT 4 Case File...**

Select the **FLUENT 4 Case File...** menu item to invoke the **Select File** dialog box. Specify the name of the FLUENT 4 case file to be read. ANSYS FLUENT reads *only* mesh information and zone types from the FLUENT 4 case file. You must specify boundary and cell zone conditions, model parameters, material properties, and other information after reading this file. For information about importing FLUENT 4 case files, see *FLUENT 4 Case Files* (p. 156).

4.12.19. PreBFC Files

You can read a PreBFC structured mesh file into ANSYS FLUENT using the **File/Import/PreBFC File...** menu item.

File → **Import** → **PreBFC File...**

Select the **PreBFC File...** menu item to invoke the **Select File** dialog box. Specify the name of the PreBFC structured mesh file to be read. The solver reads mesh information and zone types from the PreBFC mesh file. For information about importing PreBFC mesh files, see *PreBFC Mesh Files* (p. 149).

4.12.20. Partition Files

To perform METIS partitioning on an unpartitioned mesh, use the **File/Import/Partition/Metis...** menu item.

File → **Import** → **Partition** → **Metis...**

You may also partition each cell zone individually, using the **File/Import/Partition/Metis Zone...** menu item.

File → **Import** → **Partition** → **Metis Zone...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See *Using the Partition Filter* (p. 1622) for detailed information about partitioning.

4.12.21. CHEMKIN Mechanism

To import a CHEMKIN format, you can import the mechanism file into ANSYS FLUENT using the **File/Import/CHEMKIN Mechanism...** menu item (*Figure 16.11* (p. 827)).

File → **Import** → **CHEMKIN Mechanism...**

See *Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827) for detailed information on importing a CHEMKIN Mechanism file.

4.13. Exporting Solution Data

The current release of ANSYS FLUENT allows you to export data to ABAQUS, Mechanical APDL, Mechanical APDL Input, ASCII, AVS, ANSYS CFD-Post, CGNS, Data Explorer, EnSight, FAST, FIELDVIEW, I-deas, NASTRAN, PATRAN, RadTherm, and Tecplot formats. *Exporting Solution Data after a Calculation* (p. 85) explains how to export solution data in these formats after the calculation is complete, and *ABAQUS Files* (p. 87) to *Tecplot Files* (p. 96) provide specific information for each type of **File Type**. For information about exporting solution data during transient flow solutions, see *Exporting Data During a Transient Calculation* (p. 98).

For NASTRAN, ABAQUS, Mechanical APDL Input, I-deas Universal, and PATRAN file formats, the following quantities are exported [*3*] (p. 2209):

- Nodes, Elements
- Node Sets (Boundary Conditions)
- Temperature
- Pressure
- Heat Flux
- Heat Transfer Coefficient
- Force

  To generate the force data that is exported for nodes at boundaries, ANSYS FLUENT performs the following steps:

  1. Facial force for each wall face is calculated by summing the pressure force, viscous force and surface tension force of the face.
  2. Partial force for each wall face is calculated by dividing its facial force by its number of shared nodes.
  3. Total force for each wall node is calculated by summing the partial forces of all the wall faces sharing that node.

For additional information, please see the following section:
4.13.1. Exporting Limitations

4.13.1. Exporting Limitations

Note the following limitations when exporting solution data:

- When using the parallel version of ANSYS FLUENT, you can only export to the following packages:
  - ABAQUS

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - ANSYS CFD-Post
  - ASCII
  - CGNS
  - EnSight Case Gold
  - Fieldview Unstructured
  - I-deas Universal
  - Mechanical APDL Input
  - NASTRAN
  - PATRAN
  - Tecplot

  The exported file will be written by the host process (see *Introduction to Parallel Processing* (p. 1585)). Note that the memory required to write the file may exceed the memory available to the host process.

- When using the parallel version of ANSYS FLUENT, the only packages to which you can export custom field functions are the following:
  - ANSYS CFD-Post
  - EnSight Case Gold
  - Fieldview Unstructured

  For further information about custom field functions, see *Custom Field Functions* (p. 1579).

- ANSYS FLUENT cannot import surfaces. Consequently, if you export a file from ANSYS FLUENT with surfaces selected, you may not be able to read these files back into ANSYS FLUENT. However, TGrid can import surface data (For details, see the TGrid User's Guide for details).
- ANSYS FLUENT supports exporting polyhedral data only for ASCII, EnSight Case Gold, and Fieldview Unstructured file formats. For further details, see *ASCII Files* (p. 89), *EnSight Case Gold Files* (p. 91), and *Fieldview Unstructured Files* (p. 94).
- If the files that are exported during multiple transient simulations are to be used as a set, you must make sure that all of the simulations are run on the same platform, using the same number of processors. This ensures that all of the files are compatible with each other.

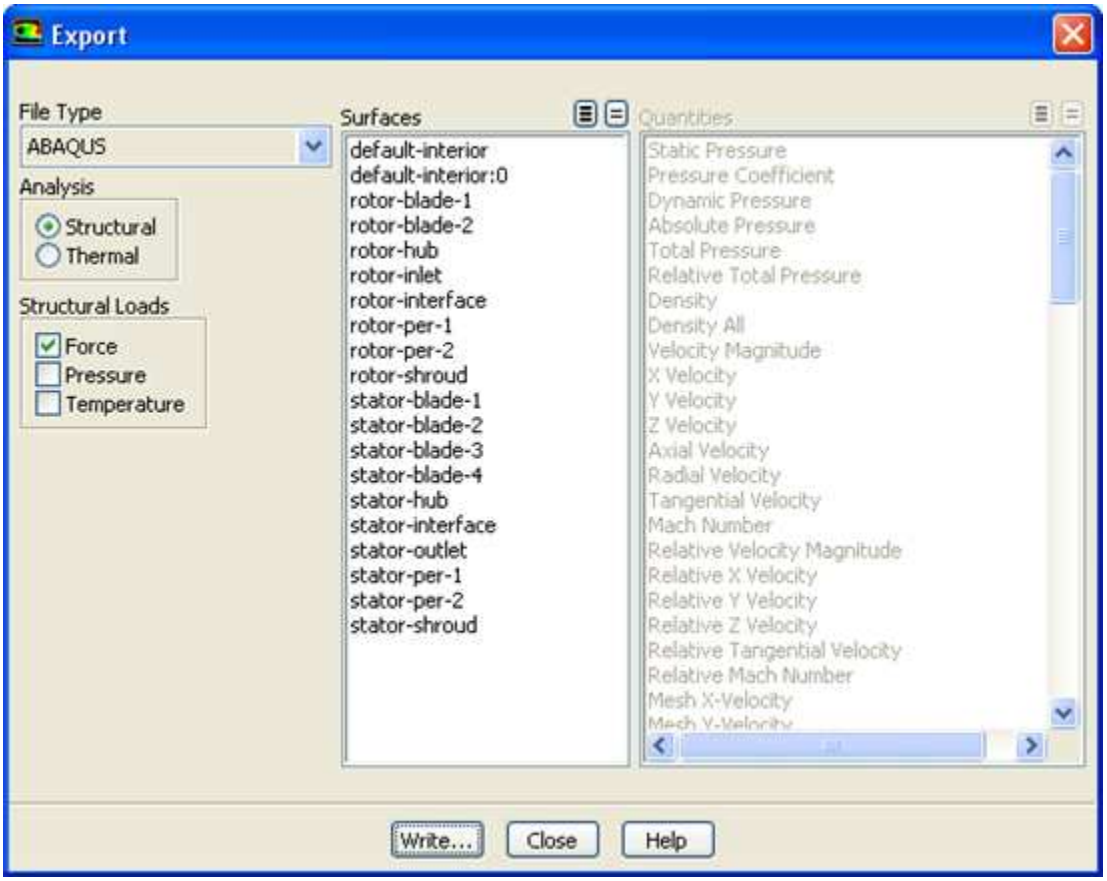
4.14. Exporting Solution Data after a Calculation

To export solution data to a different file format after a calculation is complete, use the *Export Dialog Box* (p. 2056) (*Figure 4.5* (p. 86)).

File → **Export** → **Solution Data...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.5 The Export Dialog Box



Information concerning the necessary steps and available options for each **File Type** are listed in *ABAQUS Files* (p. 87) to *Tecplot Files* (p. 96).

For details about general limitations for exporting solution data and the manner in which it is exported, see *Exporting Solution Data* (p. 84).

For additional information, please see the following sections:

4.14.1. ABAQUS Files
4.14.2. Mechanical APDL Files
4.14.3. Mechanical APDL Input Files
4.14.4. ASCII Files
4.14.5. AVS Files
4.14.6. ANSYS CFD-Post-Compatible Files
4.14.7. CGNS Files
4.14.8. Data Explorer Files
4.14.9. EnSight Case Gold Files
4.14.10. FAST Files
4.14.11. FAST Solution Files
4.14.12. Fieldview Unstructured Files
4.14.13. I-deas Universal Files
4.14.14. NASTRAN Files
4.14.15. PATRAN Files
4.14.16. RadTherm Files
4.14.17. Tecplot Files

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.14.1. ABAQUS Files

Select ABAQUS from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

When the **Energy Equation** is enabled in the *Energy Dialog Box* (p. 1645), you can choose the loads to be written based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**. Note the following limitations with these loads:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (i.e., if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. The exported file format of ABAQUS (`file.inp`) contains coordinates, connectivity, zone groups, and optional loads.

Export of data to ABAQUS is valid only for solid zones or for those surfaces that lie at the intersection of solid zones. Temperature data is exported for the whole domain.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.2. Mechanical APDL Files

Export to Mechanical APDL can only be invoked using the `file/export/mechanical-apdl` text command. You will be prompted to enter a `Mechanical APDL file name` and the `Zone to Export`.

The file written is a Mechanical APDL results file with a `.rfl` extension. This file preserves the cell zones defined in ANSYS FLUENT. The Mechanical APDL file is a single file containing coordinates, connectivity, and the scalars listed below:

```
''x-velocity'', ''y-velocity'', ''z-velocity'', ''pressure'',
''temperature'', ''turb-kinetic-energy'', ''turb-diss-rate'',
''density'', ''viscosity-turb'', ''viscosity-lam'', ''viscosity-eff'',
''thermal-conductivity-lam'', ''thermal-conductivity-eff'',
''total-pressure'', ''total-temperature'', ''pressure-coefficient'',
''mach-number'', ''stream-function'', ''heat-flux'',
''heat-transfer-coef'', ''wall-shear'', ''specific-heat-cp''
```

Important

Export to ANSYS is available on a limited number of platforms (alpha, hpux11, hpux11_64, solaris, ultra, aix51, power3, irix65_mips4, lnx86, and ntx86).

To read this file into Mechanical APDL, do the following:

1. In Mechanical APDL, expand the **General Postproc** item in the **ANSYS Main Menu**, and click **Data & File Opts**. Then perform the following steps in the **Data and File Options** dialog box that opens.
   a. Make sure that **Read single result file** is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

   b. Click the browse button to open the **Open** dialog box, and select the `.rfl` file generated by ANSYS FLUENT.
   c. Click **Open** in the **Open** dialog box.
   d. Click **OK** in the **Data and File Options** dialog box.
2. In the **ANSYS Main Menu**, click **Results Summary** in the expanded **General Postproc** list. A **SET,LIST Command** window will open, displaying a summary of the file contents.
3. In the expanded **General Postproc** list, expand the **Read Results** item and click **First Set** to read the data from the file.
4. Display the data from the file. You can display mesh information by selecting either the **Plot/Nodes** menu item or the **Plot/Elements** menu item from the Mechanical APDL menu bar. You can display results by selecting either the **Plot/Results/Contour Plot/Nodal Solution...** menu item or the **Plot/Results/Contour Plot/Elem Solution...** menu item, and then using the dialog box that opens.

You have the option of using the **Execute Commands** dialog box to export data to Mechanical APDL at specified intervals during the calculation. The text command for exporting can be entered directly into the **Command** text box (or the ANSYS FLUENT console, if you are defining a macro). It is of the following form:

`file/export/mechanical-apdl` *file_name list_of_cell_zones* `()`

where

- *file_name* specifies the name (without the extension) of the file that you wish to write. You should include a special character string in the file name, so that the solver assigns a new name to each file it saves. For details, see *Automatic Numbering of Files* (p. 61). By using a special character string, you can have the files numbered by time step, flow time, etc.
- *list_of_cell_zones* specifies the list of cell zones (separated by commas) from which you want to export data. The `()` input terminates the list. For example, the input `fluid-rotor, fluid-stator()` will select the cell zones named **fluid-rotor** and **fluid-stator**.

See *Executing Commands During the Calculation* (p. 1288) for information about executing commands and creating and using command macros.

4.14.3. Mechanical APDL Input Files

Select Mechanical APDL Input from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

When the **Energy Equation** is enabled in the *Energy Dialog Box* (p. 1645), you can choose the loads to be written based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**. Note the following limitations with these loads:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (i.e., if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. ANSYS FLUENT exports an input file that contains Mechanical APDL finite element information including nodes, elements, and loads that can

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be used to do finite element analysis in Mechanical APDL with minimal effort. The file format is written in `.cdb` format. The export of Mechanical APDL Input files is in ASCII format and thus is available on all platforms.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.4. ASCII Files

Select **ASCII** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. Also select the variable(s) for which data is to be saved in the **Quantities** list.

When exporting ASCII files, you have the following options:

- Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.
- Select the **Delimiter** separating the fields (**Comma** or **Space**).

Click the **Write...** button to save the file, using the **Select File** dialog box. ANSYS FLUENT will export a single ASCII file containing coordinates, optional loads, and specified scalar function data.

Important

ANSYS FLUENT supports exporting polyhedral data to ASCII.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.5. AVS Files

Select **AVS** from the **File Type** drop-down list and specify the scalars you want in the **Quantities** list.

Click the **Write...** button to save the file, using the **Select File** dialog box. An AVS version 4 UCD file contains coordinate and connectivity information and specified scalar function data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.6. ANSYS CFD-Post-Compatible Files

To export data to a file that is compatible with ANSYS CFD-Post, select **CFD-Post Compatible** from the **File Type** drop-down list and select the variables for which you want data saved from the **Quantities** selection list.

Specify the format of the file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported data into ANSYS CFD-Post and requires less storage space.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file (e.g., `file.cdat`) is written, containing the specified variable data for the entire domain.

Important

The `.cdat` file should be saved in the same folder as the case file which produced the data. Then when you read the `.cdat` file into ANSYS CFD-Post, the associated case file will also be read in automatically. If they are not saved in the same folder, ANSYS CFD-Post will prompt you to specify the appropriate case file.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.7. CGNS Files

Select **CGNS** from the **File Type** drop-down list and specify the scalars you want in the **Quantities** list.

Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.

Click the **Write...** button to save the file, using the **Select File** dialog box. CGNS (CFD general notation system) is a single file (e.g., `file.cgns`) containing coordinates, connectivity, zone information, velocity, and selected scalars.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.8. Data Explorer Files

Select **Data Explorer** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. Also specify the scalars you want in the **Quantities** list.

Important

When you are exporting data for Data Explorer, EnSight Case Gold, or I-deas Universal and the reference zone is not a stationary zone, the data in the velocity fields is exported by default as velocities relative to the motion specification of that zone. This data is always exported, even if you do not choose to export any scalars. Any velocities that you select to export as scalars in the **Quantities** list (e.g., **X Velocity**, **Y Velocity**, **Radial Velocity**, etc.) are exported as absolute velocities. For all other types of exported files, the velocities exported by default are absolute velocities.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file (e.g., `file.dx`) is exported, containing coordinate, connectivity, velocity, and specified function data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.14.9. EnSight Case Gold Files

When exporting to EnSight, you can choose to export the data associated with a single data file, or you can export the data associated with multiple files that were generated by a transient solution.

- To export the solution data from a single data file, read in the case and data file and open the **Export** dialog box.

  File → **Export** → **Solution Data...**

  Then perform the following steps in the **Export** dialog box:

  1. Select **EnSight Case Gold** from the **File Type** drop-down list.
  2. Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.
  3. Specify the format of the file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported files into EnSight.
  4. (optional) Select the **Interior Zone Surfaces** from which you want data exported. By default, the data being exported is taken from the entire ANSYS FLUENT domain. The **Interior Zone Surfaces** selection list allows you to also specify that the data be taken from selected zone surfaces whose **Type** is identified as **interior** in the **Boundary Conditions** task page.
  5. Select the scalars you want to write from the **Quantities** selection list.
  6. Click the **Write...** button to save the file for the specified function(s), using the **Select File** dialog box.

- To export solution data associated with multiple files that were generated by a transient solution, you must first make sure that the following criteria are met:

  – All of the relevant case and data files must be in the working folder.

  – The data files must be separated by a consistent number of time steps.

  Next, enter the following text command in the console:

  `file` → `transient-export` → `ensight-gold-from-existing-files`

  Then, enter responses to the following prompts in the console:

  1. `EnSight Case Gold file name`

     Enter the name you want assigned to the exported files.

  2. `Case/Datafileselectionbybase name?`

     Enter `yes` if the case and data files share a "base name" (i.e., a common initial string of characters). The alphanumeric order of the full names must correspond to the order in which the files were created. You will then be prompted to enter the base name at the `Case / Data file base name` prompt that follows. For example, with a set of files named `elbow-0001`, `elbow-0002`, `elbow-0003`, etc., enter `elbow-` for the base name.

     Enter `no` if you have created an ASCII file in the working folder that lists the names of the data files in order of when they were created. The file should list one data file name per line. You will then be prompted to enter the name of this file at the `Provide the file name which`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`contains the data file names` prompt that follows. Note that you must include the file extension if any in your entry at the prompt.

3. `Specify Skip Value`

   Enter an integer value to specify the number of files you want to skip in between exporting files from the sequence. For example, enter `1` to export every other file, enter `2` to export every third file, etc.

4. `Cell-Centered?`

   Enter `yes` if you want to export the data values from the cell centers (or face center values, for boundary faces).

   Enter `no` if you want to export the data values from the node points.

5. `Write separate file for each time step for each variable?`

   Enter `yes` if you want separate EnSight Case Gold files written for each time step. Otherwise, all of the data for the `.scl1` and `.vel` files will be combined into a single file for each.

6. `Write in binary format?`

   Enter `yes` to write the files in binary format. Otherwise, they will be written in ASCII format. The advantage of the binary format is that it takes less time to load the exported files into EnSight.

7. `Specify Data File Frequency`

   Enter the number of time steps between the data files being exported.

8. `Separate case file for each time step?`

   Enter `no` if all the data files were generated from the same case file (i.e., the simulation involved a static mesh). Note that the name of the case file must be the same (not including the extension) as the name of the first data file in the sequence.

   Enter `yes` if the data files were generated from the different case files (i.e., the simulation involved a sliding or dynamic mesh). Note that the names of the case files must be the same (not including the extension) as the names of the corresponding data files.

9. `Read the case file?`

   Enter `no` if the first (or only, for a static mesh) case file is already in memory.

   Enter `yes` if the first (or only, for a static mesh) case file is *not* already in memory.

10. `Interior Zone Surfaces(1)`

    Enter the name of any interior zone surface from which you want data exported. By default, the data being exported is taken from the entire ANSYS FLUENT domain. This prompt allows you to also specify that the data be taken from selected zone surfaces whose **Type** is identified as **interior** in the **Boundary Conditions** task page. After you enter the first interior zone surface, you will be prompted to enter the second one, and so on, until you press `Enter` without typing any characters.

11. `EnSight Case Gold scalar(1) else q to continue`

    Enter in the first scalar quantity you want exported. You can press the `<Enter>` key to print a list of available scalar quantities in the console. After you enter the first quantity, you will be prompted

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to enter the second quantity, and so on, until you enter `q`. The EnSight Case Gold files will then be written.

When exporting to EnSight Case Gold, files will be created with the following four formats:

- A geometry file (e.g., `file.geo`) containing the coordinates and connectivity information.
- A velocity file (e.g., `file.vel`) containing the velocity.
- A scalar file (e.g., `file.scl1`) for each selected variable or function.
- An EnSight case file (e.g., `file.encas`) that contains details about the other exported files.

Important

- For non-stationary reference zones, all the velocities are exported to EnSight as velocities relative to the selected reference zone. See the informational note in *Data Explorer Files* (p. 90) for further details.
- ANSYS FLUENT supports exporting polyhedral data to EnSight.

You also have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see *Exporting Data During a Transient Calculation* (p. 98).

4.14.10. FAST Files

This file type is valid only for triangular and tetrahedral meshes. Select **FAST** from the **File Type** drop-down list and select the scalars you want to write in the **Quantities** list.

Click the **Write...** button to save the file for the specified function(s), using the **Select File** dialog box. The following files are written:

- A mesh file in extended Plot3D format containing coordinates and connectivity.
- A velocity file containing the velocity.
- A scalar file for each selected variable or function.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.11. FAST Solution Files

This file type is valid only for triangular and tetrahedral meshes. Select **FAST Solution** from the **File Type** drop-down list and click the **Write...** button. A single file is written containing density, velocity, and total energy data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.14.12. Fieldview Unstructured Files

Select **Fieldview Unstructured** from the **File Type** drop-down list and select the scalars you want to write in the **Quantities** list. Click the **Write...** button to save the file for the specified function(s), using the **Select File** dialog box.

The following files are written:

- A binary file (e.g., `file.fvuns`) containing coordinate and connectivity information and specified scalar function data.
- A regions file (e.g., `file.fvuns.fvreg`) containing information about the cell zones and the frame of reference.

The cell zone information includes the names of the cell zones along with the mesh numbers. For the moving frame of reference, the regions file contains information about the origin, the axis of rotation and the rotation speed. Volume data is written using the absolute frame of reference.

If you are running multiple steady-state solutions on the same mesh, you can export only the data files and avoid the repeated writing of the mesh file by using the following TUI command:

`file/export/fieldview-unstruct-data` *file_name list_of_scalars* `q`

where

- *file_name* specifies the name (without the extension) of the file that you wish to write.
- *list_of_scalars* specifies the list of cell functions (separated by spaces, i.e., no commas) that you want to write to the exported file. The `q` input terminates the list. For example, the input `x-velocity cell-zone q` will select x velocity and the cell volume.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.13. I-deas Universal Files

Important

If you intend to export data to I-deas, ensure that the mesh does not contain pyramidal elements, as these are currently not supported by I-deas.

Select **I-deas Universal** from the **File Type** drop-down list. Select the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enakbled in the *Energy Dialog Box* (p. 1645), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

These loads have the following limitations :

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (i.e., if no **Surfaces** are selected).

Important

For non-stationary reference zones, all the velocities are exported to I-deas Universal as velocities relative to the selected reference zone. See the informational note in *Data Explorer Files* (p. 90) for further details.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file is written containing coordinates, connectivity, optional loads, zone groups, velocity, and selected scalars.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.14. NASTRAN Files

Select **NASTRAN** from the **File Type** drop-down list. Select the surface(s) for which you want to write data in the **Surfaces** list. If you do not select any surfaces, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enabled in the *Energy Dialog Box* (p. 1645), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

These loads have the following limitations :

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (i.e., if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file (e.g., `file.bdf`) is written containing coordinates, connectivity, optional loads, zone groups, and velocity. Pressure is written as PLOAD4, and heat flux is written as QHBDYE data. If you select wall zones in the **Surfaces** list, nodal forces are written for the walls. When data is written for the heat transfer coefficient, it is based on the wall faces rather than the nodes.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see *Exporting Data During a Transient Calculation* (p. 98).

4.14.15. PATRAN Files

Select **PATRAN** from the **File Type** drop-down list. Select the surface(s) for which you want to write data in the **Surfaces** list. If you do not select any surfaces, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enabled in the *Energy Dialog Box* (p. 1645), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

These loads have the following limitations :

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (i.e., if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. A neutral file (e.g., `file.out`) is written containing coordinates, connectivity, optional loads, zone groups, velocity, and selected scalars. Pressure is written as a distributed load. If wall zones are selected in the **Surfaces** list, nodal forces are written for the walls. The PATRAN result template file (e.g., `file.res_tmpl`) is written, which lists the scalars present in the nodal result file (e.g., `file.rst`).

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see *Exporting Data During a Transient Calculation* (p. 98).

4.14.16. RadTherm Files

The option to export a RadTherm file type is available only when the **Energy Equation** is enabled in the *Energy Dialog Box* (p. 1645). Select **RadTherm** from the **File Type** drop-down list and select the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

Select the method of writing the heat transfer coefficient (**Heat Transfer Coefficient**), which can be **Flux Based** or, if a turbulence model is enabled, **Wall Function** based.

Click the **Write...** button to save the file, using the **Select File** dialog box. A PATRAN neutral file (e.g., `file.neu`) is written containing element velocity components (i.e., the element that is just touching the wall), heat transfer coefficients, and temperatures of the wall for any selected wall surface. If the wall is one-sided, the data is written for one side of the wall. If the wall is two-sided (a wall-wall shadow pair), the values are written only for the original wall face, not for the shadow face (which is a duplicate).

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.14.17. Tecplot Files

Select **Tecplot** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the data is written for the entire domain. Select the variable(s) for which data is to be saved in the **Quantities** list.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file is written containing the coordinates and scalar functions in the appropriate tabular format.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- ANSYS FLUENT exports Tecplot files in **FEBLOCK** format. The **utility fe2ram** can import Tecplot files only in **FEPOINT** format.
- If you intend to postprocess ANSYS FLUENT data with Tecplot, you can either export data from ANSYS FLUENT and import it into Tecplot, or use the Tecplot ANSYS FLUENT Data Loader included with your Tecplot distribution. The data loader reads native ANSYS FLUENT case and data files directly. If you are interested in this option, contact Tecplot, Inc. for assistance or visit www.tecplot.com.

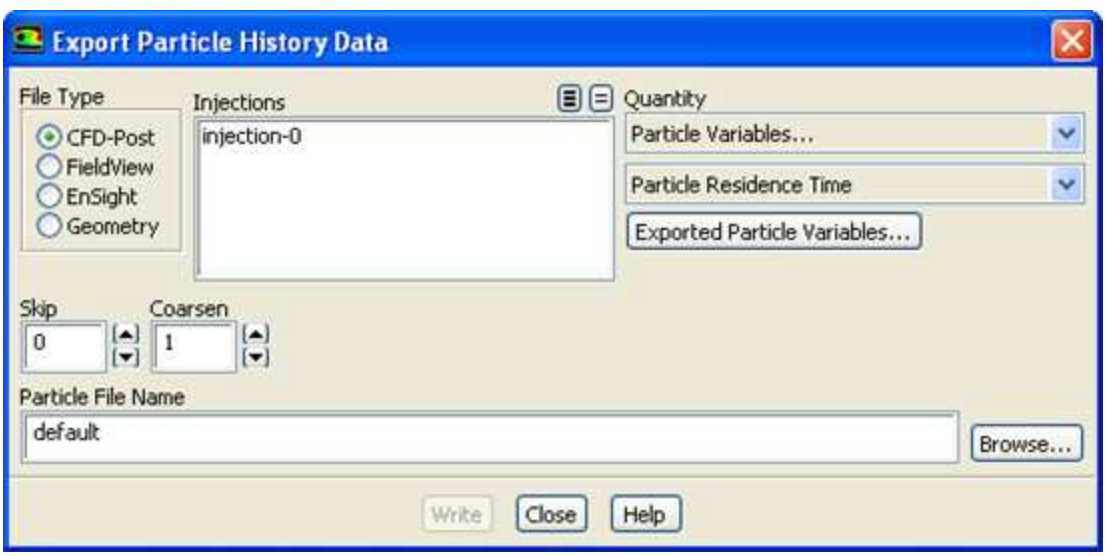
You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See *Exporting Data During a Transient Calculation* (p. 98) for the complete details.

4.15. Exporting Steady-State Particle History Data

Particle history data can be exported for steady-state solutions or for single transient particle steps by selecting the **Particle History Data...** option under the **File/Export** menu and performing the steps described in this section. For details about exporting particle history data automatically during transient simulations, see *Creating Automatic Export Definitions for Transient Particle History Data* (p. 102)

File → **Export** → **Particle History Data...**

Figure 4.6 The Export Particle History Data Dialog Box



1. Specify the **File Type** you want to export by selecting one of the following:
   - **CFD-Post** for the CFD-Post compatible format
   - **FieldView** for the FIELDVIEW format
   - **EnSight** for the EnSight format
   - **Geometry** for the `.ibl` format (not available when **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you plan to export particle data to EnSight, you should first verify that you have already written the files associated with the EnSight Case Gold file type by using the **File/Export/Solution Data...** menu option (see *EnSight Case Gold Files* (p. 91)).

Select the predefined injections that are the source of the particles from the **Injections** selection list. See *Creating and Modifying Injections* (p. 1038) for details about creating injections.

2. Select the particle variables contained in the export file by clicking the **Exported Particle Variables...** button and selecting the variables appearing in the **Reporting Variables** dialog box (*Figure 25.29* (p. 1079)), as described in *Reporting of Current Positions for Unsteady Tracking* (p. 1080)
3. If you have added the **Color by** variable in the **Reporting Variables** dialog, select an appropriate category and variable under **Quantity** for the particle data to be exported.
4. If your exported particle history file is too large to postprocess because there are too many tracks or particles written to the file, you can reduce the number of particle tracks by increasing the **Skip** value.
5. To control the exported file size, the number of points of the particle trajectories can be reduced using the **Coarsen** value. This is only valid for steady-state particle trajectories.
6. Enter the name (and folder path, if you do not want it to be written in the current folder) for the exported particle data file in the **Particle File Name** text box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.
7. If you selected **EnSight** under **File Type**, you should specify the **EnSight Encas File Name**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export/Solution Data...** menu option. The selected file will be modified and renamed as a new file that contains information about all of the related particle files that are generated during the export process (including geometry, velocity, scalars, particle and particle scalar files).

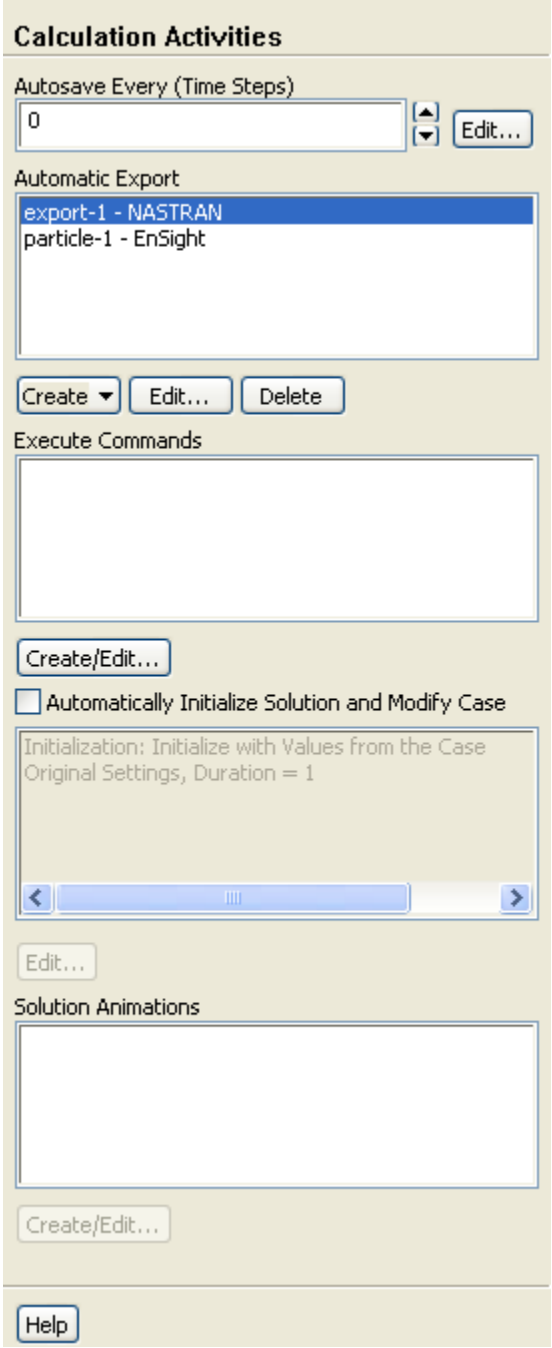
   The name of the new file will be the root of the original file with `.new` appended to it (e.g., if `test.encas` is selected, a file named `test.new.encas` will be written). It is this new file that should be read into EnSight. If you do not specify a **EnSight Encas File Name**, then you will need to create an appropriate `.encas` file manually.
8. If you selected **EnSight** under **File Type**, and you are exporting steady-state particle tracks, enter the **Number of Particle Time Steps**.
9. Click **Write** to export the particle history data. If you selected **EnSight** under **File Type**, data files will be written in both `.mpg` and `.mscl` formats.
10. Click **Close** to close the dialog box.

4.16. Exporting Data During a Transient Calculation

Before you run a transient flow solution, you can set up the case file so that solution data and particle history data is exported as the calculation progresses. This is accomplished by creating automatic export definitions using the *Calculation Activities Task Page* (p. 1942) (*Figure 4.7* (p. 99)), as described in the following sections.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.7 The Calculation Activities Task Page



The names of the automatic export definitions you create are displayed in the **Automatic Export** selection list, along with the format in which it will be exported. You can edit or delete the definition by selecting a definition in the list and clicking the **Edit...** or **Delete** button,.

For additional information, please see the following sections:

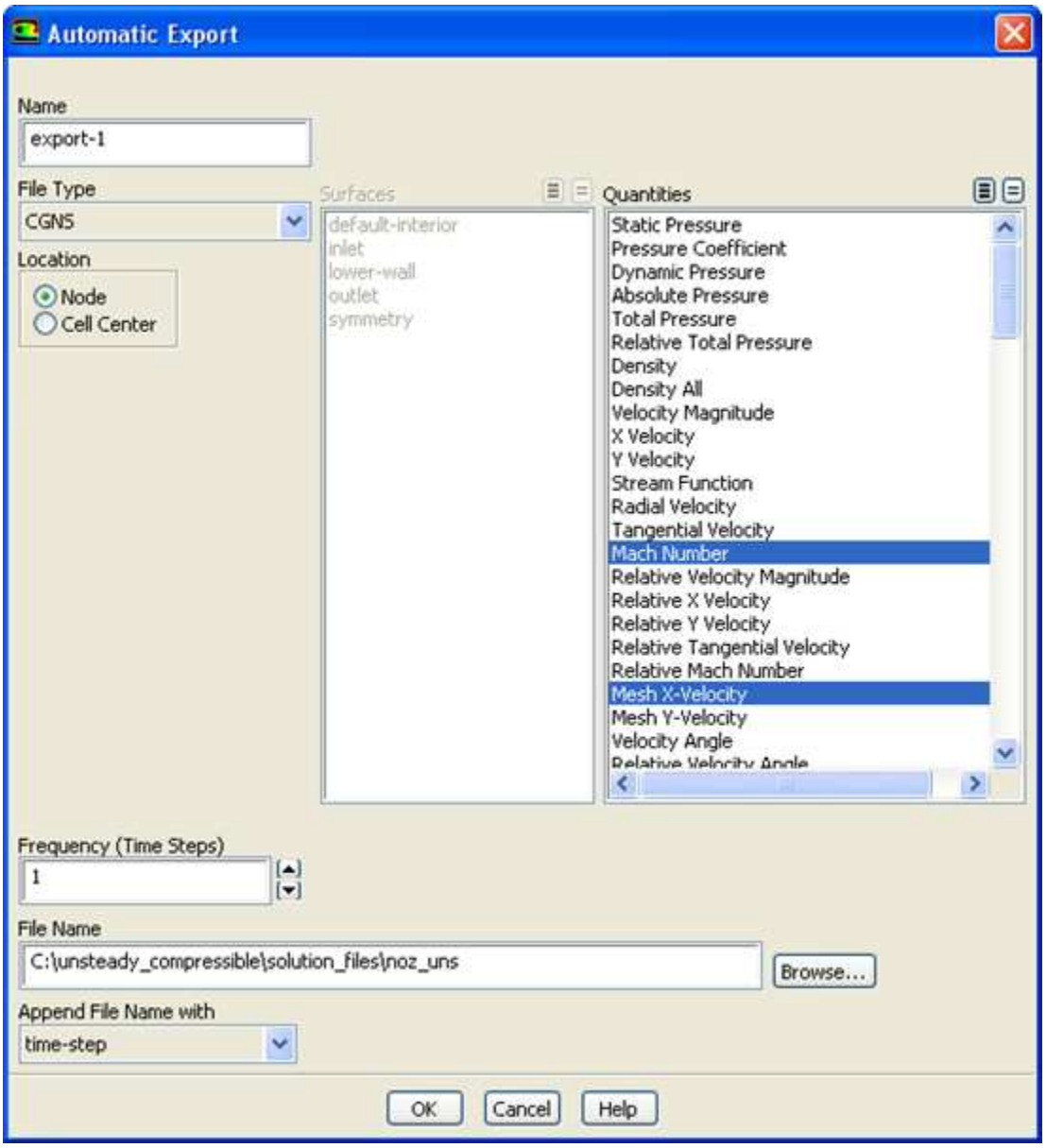
4.16.1. Creating Automatic Export Definitions for Solution Data
4.16.2. Creating Automatic Export Definitions for Transient Particle History Data

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.16.1. Creating Automatic Export Definitions for Solution Data

To create an automatic export definition for solution data, begin by making sure that **Transient** is selected for **Time** in the *General Task Page* (p. 1631). Next, click the **Create** button under the **Automatic Export** selection list in the **Calculation Activities** task page (a drop-down list will appear). Select **Solution Data Export...** from the drop-down list to open the **Automatic Export** dialog box (*Figure 4.8* (p. 100)).

Figure 4.8 The Automatic Export Dialog Box



Then perform the following steps:

1. Enter a name for the automatic export definition in the **Name** text box. This is the name that will be displayed in the **Automatic Export** selection list in the **Calculation Activities** task page.
2. Define the data to be exported by making selections in the relevant group boxes and selection lists: **File Type**, **Surfaces**, **Interior Zone Surfaces**, **Quantities**, **Analysis**, **Structural Loads**, **Thermal Loads**, **Location**, **Delimiter**, **Format**, and **Heat Transfer Coefficient**. See *ABAQUS Files* (p. 87) – *Tecplot Files* (p. 96) for details about the specific options available for the various file types.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set the **Frequency** at which the solution data will be exported during the calculation. If you enter `10` in the **Frequency** text box, for example, a file will be written after every 10 time steps.
4. If you selected **EnSight Case Gold** from the **File Type** drop-down list, the **Separate Files for Each Time Step** option allows you to specify that separate files are written at the prescribed time steps. This option is enabled by default and is the recommended practice, as it ensures that all of the data is not lost if there is a disruption to the calculation (e.g., from a network failure) before it is complete. If you choose to disable this option, all of the data for the `.scl1` and `.vel` files will be combined into a single file for each.
5. If you selected **CFD-Post Compatible** from the **File Type** drop-down list, and ANSYS FLUENT detects that the case has been modified (whether being due to the mesh being modified or due to a change in the case settings), then ANSYS FLUENT will save matching `.cas` and `.cdat` file containing the same prefix. For more information about **CFD-Post Compatible**, go to *ANSYS CFD-Post-Compatible Files* (p. 89).
6. Specify how the exported files will be named. Every file saved will begin with the characters entered in the **File Name** text box (note that a file extension is not necessary). You can specify a folder path if you do not want it written in the current folder. The **File Name** can also be specified through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   Next, make a selection in the **Append File Name with** drop-down list, to specify that the **File Name** be followed by either the time step or flow time at which it was saved. Note that this selection is not available when exporting to EnSight. When **EnSight Case Gold** is selected from the **File Type** drop-down list, the time step is always appended if the **Separate Files for Each Time Step** option is enabled; otherwise, no digits are appended.

   When appending the file name with the flow time, you can specify the number of decimal places that will be used by making an entry in the **Decimal Places in File Name** text box. By default, six decimal places will be used.
7. Click **OK** to save the settings for the automatic export definition.

For details about general limitations for exporting solution data and the manner in which it is exported, see *Exporting Solution Data* (p. 84).

Important

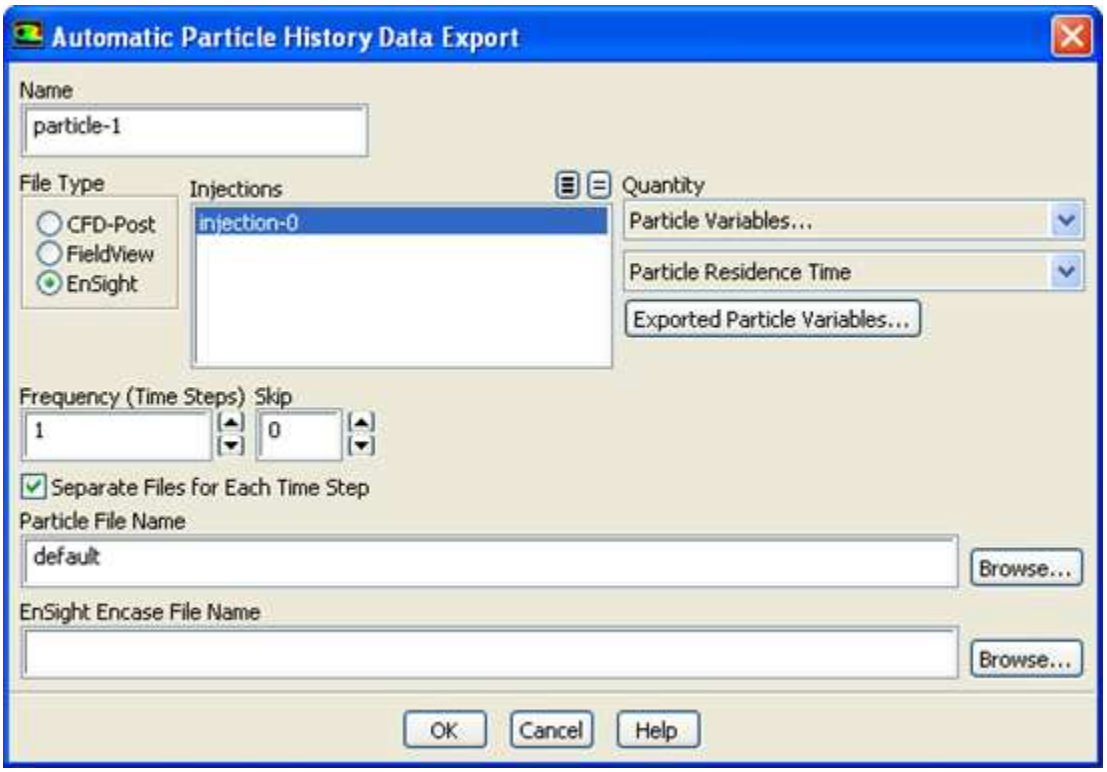
- If the files that are exported during multiple transient simulations are to be used as a set, you should run all of the simulations on the same platform, using the same number of processors. This ensures that all of the files are compatible with each other.
- If you selected **EnSight Case Gold** from the **File Type** drop-down list, note the following:
  - Though it is possible for ANSYS FLUENT to export a file that is greater than 2 Gbytes, such a file could not be read using EnSight when it is run on 32-bit Windows, as this exceeds EnSight's maximum file size.
  - ANSYS FLUENT does not support exporting data files to EnSight during a transient calculation in which a new cell zone or surface is created after the calculation has begun (as can be the case for an in-cylinder simulation, for example).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.16.2. Creating Automatic Export Definitions for Transient Particle History Data

To create an automatic export definition for particle history data, begin by making sure that **Unsteady Particle Tracking** is selected in the **Discrete Phase Model** dialog box. Next, click the **Create** button under the **Automatic Export** selection list in the **Calculation Activities** task page (a drop-down list will appear). Select **Particle History Data Export...** from the drop-down list to open the **Automatic Particle History Data Export** dialog box (*Figure 4.9* (p. 102)).

Figure 4.9 The Automatic Particle History Data Export Dialog Box



Then perform the following steps:

1. Enter a name for the automatic export definition in the **Name** text box. This is the name that will be displayed in the **Automatic Export** selection list in the **Calculation Activities** task page. Make sure that the chosen name is not already used for another **Automatic Export**.
2. Choose the **File Type** you want to export by selecting one of the following:
   - **CFD-Post** for the CFD-Post compatible format
   - **FieldView** for the FIELDVIEW format
   - **EnSight** for the EnSight format

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you plan to export particle data to EnSight, you should first set up an automatic export definition so that solution data is also exported to EnSight during this calculation (see *Creating Automatic Export Definitions for Solution Data* (p. 100)). As described in the steps that follow, some of the settings will need to correspond between the two automatic export definitions.

3. Select the predefined injections that are the source of the particles from the **Injections** selection list. See *Creating and Modifying Injections* (p. 1038) for details about creating injections.
4. Select the particle variables contained in the export file by clicking the **Exported Particle Variables...** button and selecting the variables appearing in the **Reporting Variables** dialog box (*Figure 25.29* (p. 1079)), as described in *Reporting of Current Positions for Unsteady Tracking* (p. 1080).
5. If you have added the **Color by** variable in the **Reporting Variables** dialog, select an appropriate category and variable under **Quantity** for the particle data to be exported.
6. Set the **Frequency** at which the particle history data will be exported during the calculation. If you enter `10` in the **Frequency** text box, for example, a file will be written after every 10 time steps for transient flow cases or 10 DPM Iterations for steady flow cases.
7. If your exported particle history file is too large to postprocess because there are too many tracks or particles written to the file, you can reduce the number of particle tracks by increasing the **Skip** value.
8. If you selected **EnSight Case Gold** for the **File Type**, the **Separate Files for Each Time Step** option allows you to specify that separate files are written at the prescribed time steps. This option is enabled by default and is the recommended practice, as it ensures that all of the data is not lost if there is a disruption to the calculation (e.g., from a network failure) before it is complete. If you choose to disable this option, all of the data for the `.mscl` and `.mpg` files will be combined into a single file for each.

Important

The setting for the **Separate Files for Each Time Step** option should be the same (i.e., enabled or disabled) as that of the automatic export definition you set up to export solution data to EnSight during this calculation. For details, see *Creating Automatic Export Definitions for Solution Data* (p. 100).

9. Enter the name (and folder path, if you do not want it to be written in the current folder) for the exported particle data file in the **Particle File Name** text box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button. Make sure the file name is different from other existing **Automatic Export** definitions to avoid overwriting data.
10. If you selected **EnSight** under **File Type**, you should specify the **EnSight Encas File Name**. Enter the same name (and folder path, if necessary) that you entered in the **File Name** text box when you set up the automatic export definition for exporting solution data to EnSight during this calculation. (For details, see *Creating Automatic Export Definitions for Solution Data* (p. 100)). The `.encas` file created during the solution data export will be modified and renamed as a new file that contains information about all of the related particle files that are generated after every time step during the export process (including geometry, velocity, scalars, particle and particle scalar files). The name of the new file will be the root of the original `.encas` file with `.new` appended to it (e.g., if the solution data export creates `test.encas`, a file named `test.new.encas` will be written for the particle data export). It is this new file that should be read into EnSight.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you do not specify an **EnSight Encas File Name**, you will have to manually create an appropriate `.encas` file.

Note that

11. Click **OK** to save the settings for the automatic export definition.

The particle data will be exported as it is generated during the transient calculation. If you selected **EnSight** under **File Type**, data files will be written in both `.mpg` and `.mscl` formats.

4.17. Exporting to ANSYS CFD-Post

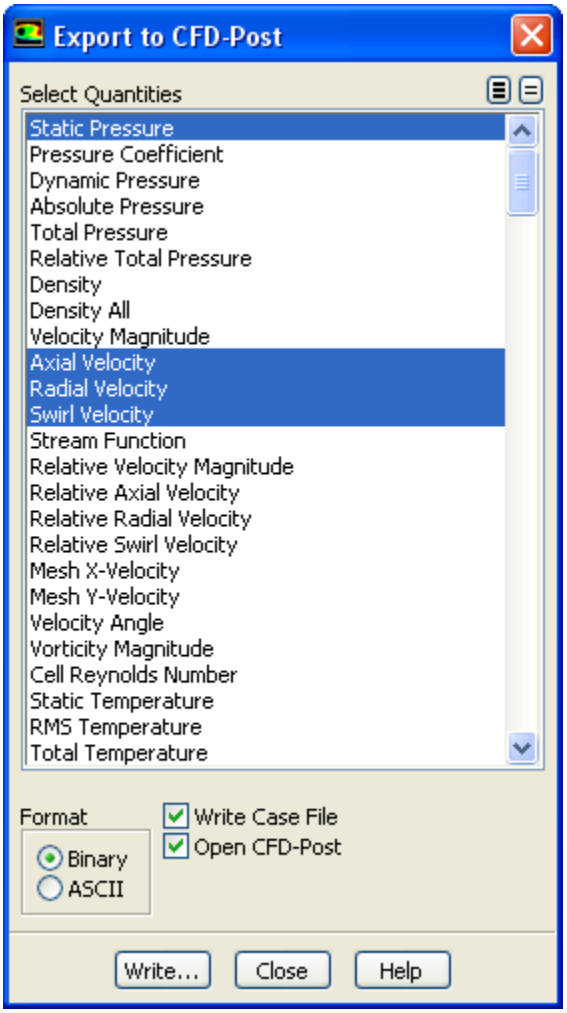
ANSYS FLUENT will allow you to send your case and data files to ANSYS CFD-Post, allowing you to perform various postprocessing activities.

File → **Export to CFD-Post...**

The **Export to CFD Post** dialog box will open (*Figure 4.10* (p. 105)), where you can select the quantities to export. You have the option to save your case file by enabling **Write Case File** and you can **Open CFD-Post**. These options are enabled by default. When writing your case file, you can choose to save it in **Binary** or **ASCII** format.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.10 The Export to CFD Post Dialog Box



Click the **Write...** button to open the **Select File** dialog box, where you will save your file.

Important

If you are saving a case file, then this is indicated in the dialog box. If you are not saving the case file, then only the `.cdat` file is written.

Clicking **OK** in the **Select File** dialog box will write the file(s) if **Write Case File** is enabled. If **Open CFD-Post** is enabled,

1. an ANSYS CFD-Post session is opened
2. the case and `.cdat` files are loaded
3. ANSYS CFD-Post displays the results

If the **Open CFD-Post** option is disabled, you can go to CFD-Post and select **Load Results**, under the **File** menu. This will load all of the results files the case is using and retain the state file. For more information about this feature in ANSYS CFD-Post, please refer to the ANSYS CFD-Post manual.

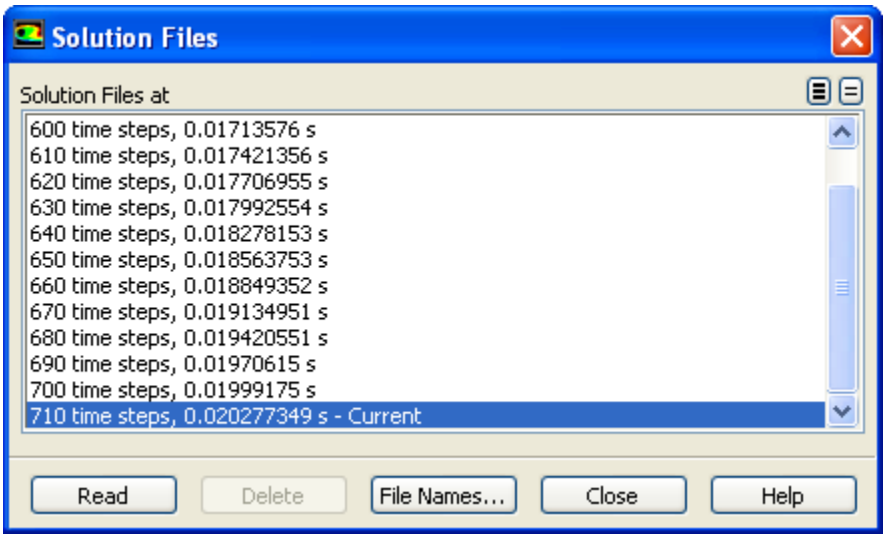
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.18. Managing Solution Files

You can manage your solution files effectively and efficiently using the **Solution Files** dialog box. Here, you can select previously saved files created using the **Autosave** dialog box and read or delete them.

File → **Solution Files...**

Figure 4.11 The Solution Files Dialog Box



The **Solution Files** dialog box (*Figure 4.11* (p. 106)) lists all of the solution files that have been automatically saved. They are listed by iteration number or time step/flow time. For the file that is currently read in, the status of `current` will appear in the **Solution Files at** list. You can make any of the files in the list current by clicking the **Read** button. Note that if more than one file is selected, the **Read** button is disabled. When an earlier solution is made current, the solution files that were generated for a later iteration/time step will be removed from this list when the calculation continues.

You can delete solution files by selecting an entry in the list and clicking **Delete**. Note that a currently loaded solution file cannot be deleted, however multiple (non-current) files can be selected and deleted. If multiple files are selected and one of those files is a currently loaded solution file, clicking **Delete** will result in the current solution file being skipped.

You can click the **File Names...** button to obtain information about the solution files and the path of the associated files.

The **Solution Files** dialog box is particularly useful for reading in files that were saved during the autosave session, since case and data files may not necessarily have the same file name.

4.19. Mesh-to-Mesh Solution Interpolation

ANSYS FLUENT can interpolate solution data for a given geometry from one mesh to another, allowing you to compute a solution using one mesh (e.g., hexahedral) and then change to another mesh (e.g., hybrid) and continue the calculation using the first solution as a starting point.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

ANSYS FLUENT does zeroth-order interpolation for interpolating the solution data from one mesh to another.

For additional information, please see the following sections:

4.19.1. Performing Mesh-to-Mesh Solution Interpolation
4.19.2. Format of the Interpolation File

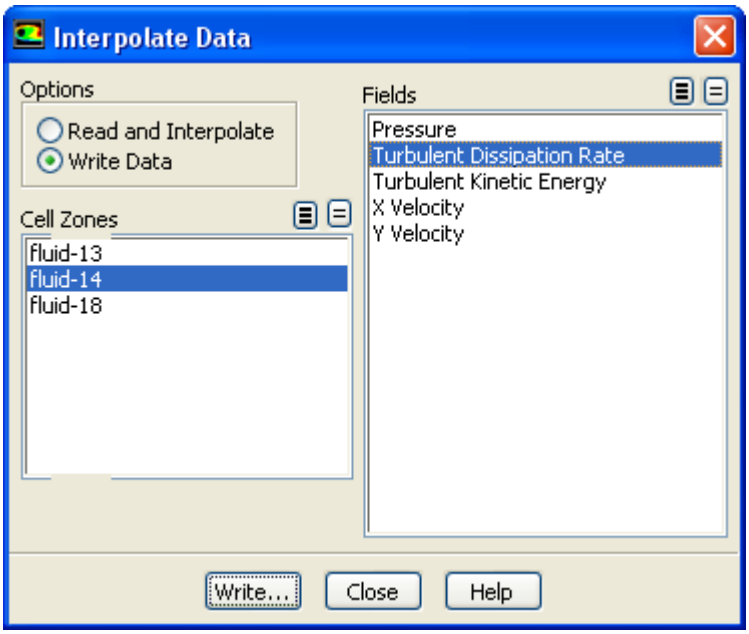
4.19.1. Performing Mesh-to-Mesh Solution Interpolation

The procedure for mesh-to-mesh solution interpolation is as follows:

1. Set up the model and calculate a solution on the initial mesh.
2. Write an interpolation file for the solution data to be interpolated onto the new mesh, using the **Interpolate Data** dialog box (*Figure 4.12* (p. 107)).

   File → **Interpolate...**

   Figure 4.12 The Interpolate Data Dialog Box



   a. Under **Options**, select **Write Data**.

   b. In the **Cell Zones** selection list, select the cell zones for which you want to save data to be interpolated.

   Note

   If your case includes both fluid and solid zones, write the data for the fluid zones and the data for the solid zones to separate files.

   c. Select the variable(s) for which you want to interpolate data in the **Fields** selection list. All ANSYS FLUENT solution variables are available for interpolation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Click **Write...** and specify the interpolation file name in the resulting **Select File** dialog box. The file format is described in *Format of the Interpolation File* (p. 108).

3. Set up a new case.

   a. Read in the new mesh, using the appropriate menu item in the **File/Read/** or **File/Import/** menu.

   b. Define the appropriate models.

   Important

   Enable all of the models that were enabled in the original case. For example, if the energy equation was enabled in the original case and you forget to enable it in the new case, the temperature data in the interpolation file will not be interpolated.

   c. Define the boundary conditions, material properties, etc.

   Important

   An alternative way to set up the new case is to save the boundary conditions from the original model using the `write-settings` text command, and then read in those boundary conditions with the new mesh using the `read-settings` text command. See *Reading and Writing Boundary Conditions* (p. 72) for further details.

4. Read in the data to be interpolated.

   File → **Interpolate...**

   a. Under **Options**, select **Read and Interpolate**.

   b. In the **Cell Zones** list, select the cell zones for which you want to read and interpolate data.

   If the solution has not been initialized, computed, or read, all zones in the **Cell Zones** list are selected by default, to ensure that no zone remains without data after the interpolation. If all zones already have data (from initialization or a previously computed or read solution), select a subset of the **Cell Zones** to read and interpolate data onto a specific zone (or zones).

   c. Click the **Read...** button and specify the interpolation file name in the resulting **Select File** dialog box.

   Important

   If your case includes both fluid and solid zones, the two sets of data are saved to separate files. Hence perform these steps twice, once to interpolate the data for the fluid zones and once to interpolate the data for the solid zones.

5. Reduce the under-relaxation factors and calculate on the new mesh for a few iterations to avoid sudden changes due to any imbalance of fluxes after interpolation. Then increase the under-relaxation factors and compute a solution on the new mesh.

4.19.2. Format of the Interpolation File

An example of an interpolation file is shown below:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
2
2
34800
3
x-velocity
pressure
y-velocity
-0.068062
-0.0680413
...
```

The format of the interpolation file is as follows:

- The first line is the interpolation file version. It is `1.0` for ANSYS FLUENT 5 and `2.0` for ANSYS FLUENT 6.
- The second line is the dimension (2 or 3).
- The third line is the total number of points.
- The fourth line is the total number of fields (temperature, pressure, etc.) included.
- Starting at the fifth line is a list of field names. To see a complete list of the field names used by ANSYS FLUENT, enter the `display/contour` text command and view the available choices by pressing `<Enter>` at the `contours of>` prompt. The list depends on the models turned on.
- After the field names is a list of x, y, and (in 3D) z coordinates for all the data points.
- At the end is a list of the field values at all the points in the same order as their names. The number of coordinate and field points should match the number given in line 3.

4.20. Mapping Electromagnetic Losses

ANSYS FLUENT allows you to perform a one way electromagnetic-thermal interaction problem by solving an electromagnetic problem in the Ansoft HFSS, Maxwell, or Q3D Extractor applications, and mapping the resulting volumetric loss information onto the solid cell zones of your ANSYS FLUENT mesh. The loss information is mapped onto the cell centroids using zeroth-order interpolation, and is considered a heat source that is added to the energy equation. This capability is useful for applications such as simulating fluid flow around or inside electromechanical devices, when the temperature of the device is influenced by electromagnetic or electrical fields.

Perform the following steps to map electromagnetic losses onto an ANSYS FLUENT mesh:

1. Set up and solve your electromagnetic problem in one of the following Ansoft applications: HFSS, Maxwell, or Q3D Extractor. For a list of the supported versions of Ansoft software, see the Release Notes.

   Important

   - If it is a HFSS simulation, the mesh must be 3D.
   - If it is a 2D Maxwell simulation, the mesh must be in either the XY or XZ plane; for an axisymmetric mesh, the z axis must be the axis of symmetry.
   - **EM Mapping** is only available in serial ANSYS FLUENT. It is not available in parallel ANSYS FLUENT.

   When the calculation is complete, export a transfer file (with the extension `.xml`) for **ANSYS**, with the link type set as **Thermal Link for ANSYS FLUENT**.

2. It is recommended that you set the environment variable `ANSOFTROOT` to be the root path for the Ansoft application.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Launch ANSYS FLUENT and read in an appropriate mesh.

Important

Note the following with regard to the ANSYS FLUENT mesh:

- The mesh should have solid cell zones that are spatially coincident with the regions of the Ansoft geometry that experience the electromagnetic losses.
- Typically, the mesh should have the same number of dimensions (i.e., 3D or 2D) as the mesh used in the Ansoft application. The only allowable exception is that you can use a 2D mesh in Maxwell with a 3D ANSYS FLUENT mesh.
- If you use Maxwell for the electromagnetic analysis, any 2D ANSYS FLUENT mesh must be in the XY plane.

4. Enable the energy equation in ANSYS FLUENT, using the **Energy** dialog box.

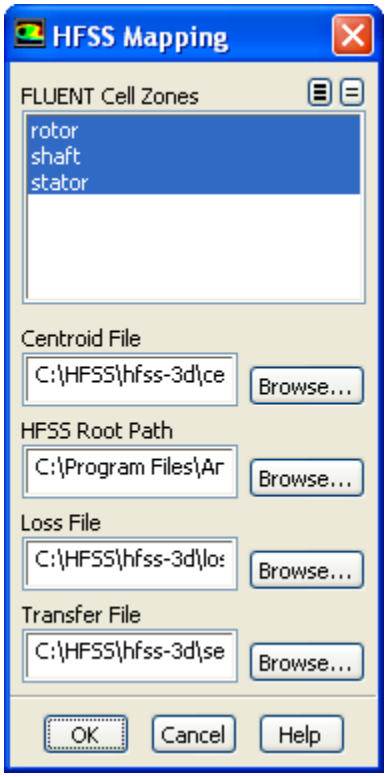
Models → **Energy** → **Edit...**

5. Use the appropriate electromagnetic mapping dialog box to map the losses, based on the Ansoft application you used to run the electromagnetic analysis.

If you used HFSS, open the **HFSS Mapping** dialog box (*Figure 4.13* (p. 110)).

File → **EM Mapping** → **HFSS...**

Figure 4.13 The HFSS Mapping Dialog Box

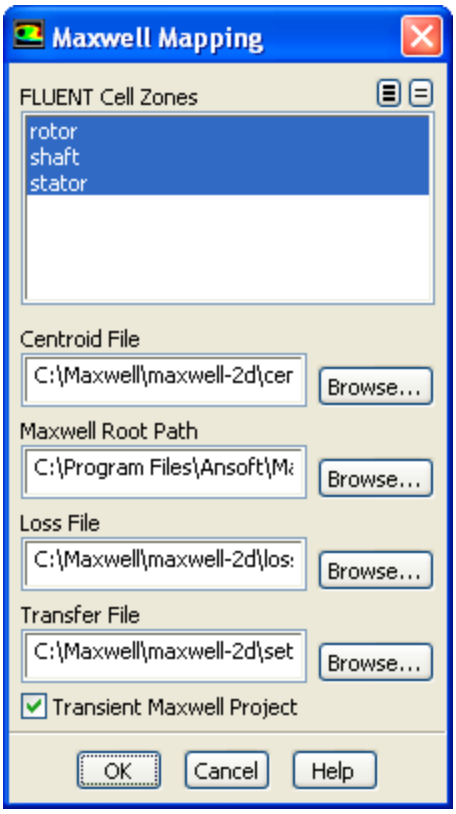


If you used Maxwell, open the **Maxwell Mapping** dialog box (*Figure 4.14* (p. 111)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → EM Mapping → Maxwell...

Figure 4.14 The Maxwell Mapping Dialog Box

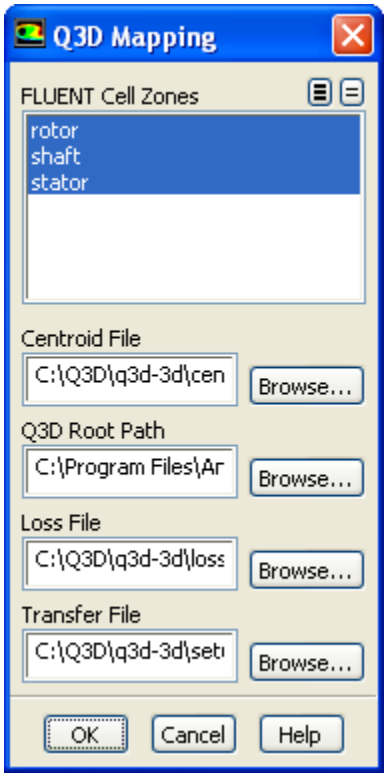


If you used Q3D Extractor, open the **Q3D Mapping** dialog box (*Figure 4.15* (p. 112)).

File → EM Mapping → Q3D...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.15 The Q3D Mapping Dialog Box



Then perform the following steps:

a. Select the solid cell zones on which you want the losses mapped from the **FLUENT Cell Zones** selection list. By default, all of the solid cell zones are selected.

b. Enter a name for **Centroid File**, which will be assigned to a `.dat` file that ANSYS FLUENT generates during the mapping process. This file will be used to provide cell centroid data to the Ansoft application.

c. Enter the root path for the Ansoft application in the **HFSS Root Path**, **Maxwell Root Path**, or **Q3D Root Path** text box. This path enables ANSYS FLUENT to launch the Ansoft application. Note that if you set the environment variable `ANSOFTROOT` in a previous step, this text box will be filled automatically.

d. Enter a name for **Loss File**, which will be assigned to a `.xml` file that the Ansoft application generates during the mapping process. This file will be used to provide the volumetric loss data to ANSYS FLUENT.

e. Enter the name of the **Transfer File** you exported from the Ansoft application in a previous step.

f. If the electromagnetic analysis was run as a steady-state project in Maxwell, disable the **Transient Maxwell Project** option.

g. Click **OK**.

ANSYS FLUENT will create the centroid file, as well as an `Input_to_Ansoft.xml` file that is used to launch the Ansoft application. Then the Ansoft application will generate the loss file, which ANSYS FLUENT will read and map to the specified solid cell zones.

6. After the loss data is mapped to the ANSYS FLUENT mesh, you can continue any further simulations in ANSYS FLUENT and/or perform postprocessing to view the results. When postprocessing, you can

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

visualize the mapped loss distribution by selecting **Temperature...** and **Temperature EM Source** from the **Contours of** drop-down lists in the **Contours** dialog box.

4.21. Mapping Data for Fluid-Structure Interaction (FSI) Applications

ANSYS FLUENT allows you to map variables (e.g., temperature, pressure) from the cell or face zones of an ANSYS FLUENT simulation onto locations associated with a finite element analysis (FEA) mesh. The results are written to a file for inclusion into an FEA simulation. During this process, both the original and the new mesh can be viewed simultaneously. ANSYS FLUENT maps the data using zeroth-order interpolation, and can write the output file in a variety of formats.

This capability is useful when solving fluid-structure interaction (FSI) problems, and allows you to perform further analysis on the solid portion of your model using FEA software. Mapping the data may be preferable to simply exporting the ANSYS FLUENT data file (as described in *Exporting Solution Data* (p. 84)), since the meshes used in CFD analysis are typically finer than those used in finite element analysis.

For additional information, please see the following sections:

4.21.1. FEA File Formats
4.21.2. Using the FSI Mapping Dialog Boxes

4.21.1. FEA File Formats

The FEA software types that are compatible with ANSYS FLUENT's FSI mapping capability include ABAQUS, I-deas, ANSYS, NASTRAN, and PATRAN. For details about the kinds of files that can be read or written during this process, see *Table 4.3: FEA File Extensions for FSI Mapping* (p. 113).

Table 4.3 FEA File Extensions for FSI Mapping

Type	Input File	Output File
ABAQUS	`.inp`	`.inp`
I-deas	`.unv`	`.unv`
ANSYS	`.cdb`, `.neu`	`.cdb`
NASTRAN	`.bdf`	`.bdf`
PATRAN	`.neu`, `.out`, `.pat`	`.out`

4.21.2. Using the FSI Mapping Dialog Boxes

To begin the process of mapping ANSYS FLUENT data, you must first create a mesh file that can be used as the input file in the steps that follow. The resolution of the mesh should be appropriate for your eventual finite element analysis. You are free to use the method and preprocessor of your choice in the creation of this file, but the end result must correspond to one of the entries in the Input File column of *Table 4.3: FEA File Extensions for FSI Mapping* (p. 113).

When creating the input file, note the following:

- While the input file may be scaled when it is read into ANSYS FLUENT, the volumes or surfaces on which the data is to be mapped must otherwise be spatially coincident with their counterparts in the ANSYS FLUENT simulation.
- The input file can be only a portion of the overall FEA model (i.e., you can exclude the parts of the model on which you are not mapping ANSYS FLUENT data). When this is the case, note that the num-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bering of the nodes and elements in the input file must match the numbering of the nodes and elements in the complete file you will use for your finite element analysis.

Next, read a case file in ANSYS FLUENT and make sure data is available for mapping, either by running the calculation or by reading a data file.

Finally, perform the following steps to generate an output file in which the ANSYS FLUENT data has been mapped to the mesh of the input file:

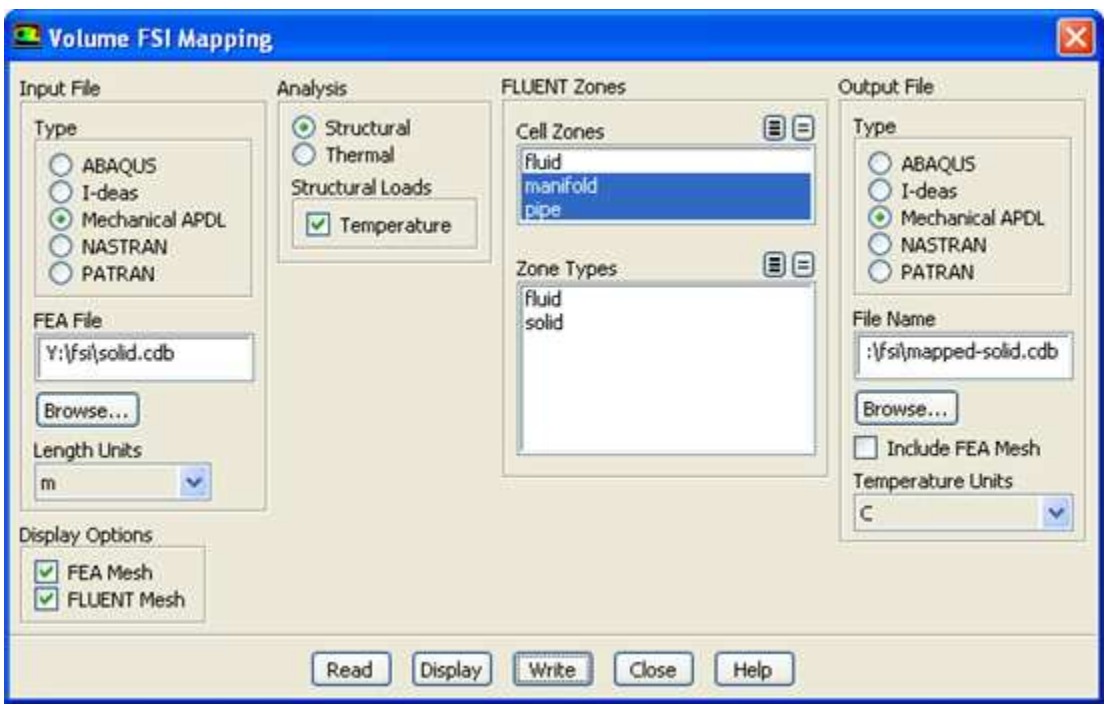
1. Open the ANSYS FLUENT dialog box that is appropriate for the zones from which the data is to be taken. If the data you are mapping is from a volume (e.g., the cell zone of a solid region), open the **Volume FSI Mapping** dialog box using the **File/FSI Mapping/Volume...** menu item (*Figure 4.16* (p. 114)). If instead the data is from a surface (e.g., a face boundary zone), open the **Surface FSI Mapping** dialog box using the **File/FSI Mapping/Surface...** menu item (*Figure 4.17* (p. 115)).

   File → **FSI Mapping** → **Volume...**

   or

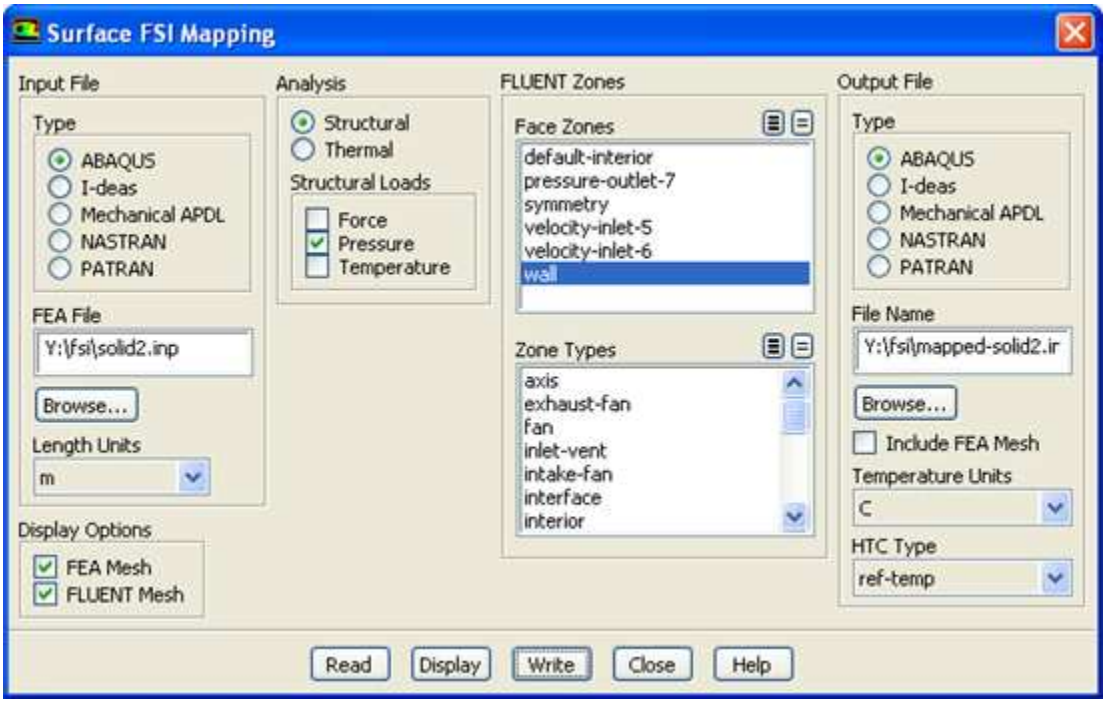
   File → **FSI Mapping** → **Surface...**

   Figure 4.16 The Volume FSI Mapping Dialog Box for Cell Zone Data



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.17 The Surface FSI Mapping Dialog Box for Face Zone Data



2. Specify the parameters of the input file and read it into ANSYS FLUENT.

   a. Select the format of the input file from the **Type** list in the **Input File** group box, based on the FEA software with which it is associated. The choices include:

      - **ABAQUS**
      - **I-deas**
      - **Mechanical APDL**
      - **NASTRAN**
      - **PATRAN**

      For a list of the file extensions associated with these types, see the Input File column of *Table 4.3: FEA File Extensions for FSI Mapping* (p. 113).

   b. Enter the name and extension (along with the folder path, if it is not in the current folder) of the input file in the **FEA File** text-entry box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   c. Specify the length units that were used in the creation of the input file by making a selection from the **Length Units** drop-down menu. This ensures that the input file is scaled appropriately relative to the ANSYS FLUENT file.

   d. Click the **Read** button to read the input file into memory.

Important

Note that the input file will only be held in memory until the output file is written, or until the FSI mapping dialog box is closed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Display the meshes so that you can visually verify that the input file is properly scaled and aligned with the ANSYS FLUENT mesh file.
   a. Make sure that the **FEA Mesh** and **FLUENT Mesh** options are enabled in the **Display Options** group box. Note that you can disable either of these options if you want to examine one of the meshes independently.
   b. Click the **Display** button to display the meshes in the graphics window.

   Important

   For the ANSYS FLUENT mesh, only the zones selected in the **FLUENT Zones** group box will be displayed—in this case, the default selections. If the default zones are not appropriate, you should redisplay the meshes after you make your zone selections in a later step.

4. Specify the type of data variables to be mapped.
   a. Select either **Structural** or **Thermal** in the **Analysis** group box. Your selection should reflect the kind of further analysis you intend to pursue, and will determine what variables are available for mapping.
   b. Enable the variables you want to map in the **Structural Loads** or **Thermal Loads** group box. When mapping volume data, you can enable only **Temperature**. When mapping surface data, you can enable **Force**, **Pressure**, and **Temperature** for structural analysis, or **Temperature**, **Heat Flux**, and **Heat Trans Coeff** for thermal analysis.

   Important

   Note that the **Energy Equation** must be enabled in the **Energy** dialog box if you want to map temperature for a structural analysis or any variable for a thermal analysis.

5. Select the zones that contain the data to be mapped in the **FLUENT Zones** group box. You can select individual zones in the **Cell Zones** or **Face Zones** selection lists, or select all zones of a particular type in the **Zone Type** selection list. If you modify the default selections, you should display the meshes again, as described previously.

   Important

   - Note that all wall zones in the **Face Zones** selection list are selected by default in the **Surface FSI Mapping** dialog box, and this includes the shadow walls created for two-sided walls. If your ANSYS FLUENT file contains a wall/shadow pair (e.g., separating a solid zone from a fluid zone), you should make sure that only the correct wall or shadow of the pair is selected.
   - Inlet zones do not have heat transfer coefficient data, and so any attempts to map this combination will be ignored.

6. Specify the parameters of the output file and write it.
   a. Select the format of the output file from the **Type** list in the **Output File** group box, based on the software with which you plan to perform your finite element analysis. The choices in this list are the same as those for the input file type. Note that you can select an output file type that is different from the input file type.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details about the file extensions associated with the various types of output files, see the Output File column of *Table 4.3: FEA File Extensions for FSI Mapping* (p. 113).

b. Enter the name (with the folder path, if appropriate) of the output file in the **File Name** text-entry box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

c. To include additional FEA information like node/element information in the exported output file, enable **Include FEA Mesh**. By default, this option is disabled and therefore, only the selected boundary condition values are exported.

d. When mapping temperature for a structural analysis or any variable for a thermal analysis, make a selection in the **Temperature Units** drop-down menu. *Table 4.4: Units Associated with the Temperature Units Drop-Down List Selections* (p. 117) shows the units for the mapped variables, depending on the **Temperature Units** selection.

Table 4.4 Units Associated with the Temperature Units Drop-Down List Selections

Temperature Units Selection	Temperature	Heat Flux	Heat Transfer Coefficient
K	K	W/m^2	W/m^2-K
C	°C	W/m^2	W/m^2- °C
F	°F	BTU/ft^2-hr	BTU/ft^2-hr- °F

e. When mapping the heat transfer coefficient for a thermal analysis, make a selection in the **HTC Type** drop-down menu to determine how the heat transfer coefficient h_{eff} is calculated.

ref-temp

calculates h_{eff} using *Equation 33–44* (p. 1573), where T_{ref} is the reference temperature defined in the *Reference Values Task Page* (p. 1898). Note that this option has the same definition as the field variable **Surface Heat Transfer Coef.**, as described in *Alphabetical Listing of Field Variables and Their Definitions* (p. 1547).

cell-temp

calculates h_{eff} using the general form of *Equation 33–44* (p. 1573), but defines T_{ref} as the temperature of the cell adjacent to the face.

wall-func-htc

calculates h_{eff} using *Equation 33–55* (p. 1578). Note that this option has the same definition as the field variable **Wall Func. Heat Tran. Coef.**, as described in *Alphabetical Listing of Field Variables and Their Definitions* (p. 1547).

f. Click **Write** to write an output file in which the ANSYS FLUENT data has been mapped to the mesh of the input file.

The input file will be released from memory when the output file is written.

4.22. Saving Picture Files

Graphic window displays can be saved in various formats (including TIFF and PostScript). There can be slight differences between pictures and the displayed graphics windows, since pictures are generated using the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

internal software renderer, while the graphics windows may utilize specialized graphics hardware for optimum performance.

Many systems provide a utility to "dump" the contents of a graphics window into a raster file. This is generally the fastest method of generating a picture (since the scene is already rendered in the graphics window), and guarantees that the picture is identical to the window.

For additional information, please see the following sections:

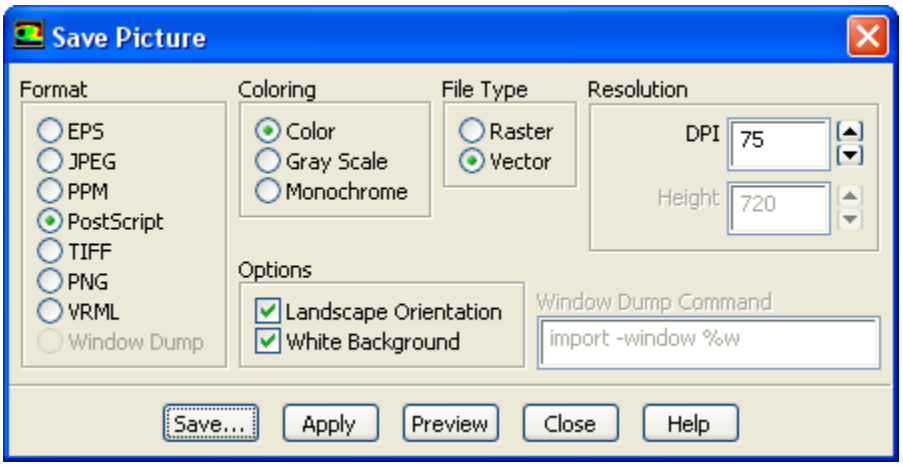
4.22.1. Using the Save Picture Dialog Box
4.22.2. Picture Options for PostScript Files

4.22.1. Using the Save Picture Dialog Box

To set picture parameters and save picture files, use the *Save Picture Dialog Box* (p. 1988)(*Figure 4.18* (p. 118)).

File → **Save Picture...**

Figure 4.18 The Save Picture Dialog Box



For your convenience, this dialog box may also be opened using the **Save Picture** button ( ) in the standard toolbar.

The procedure for saving a picture file is as follows:

1. Choose the picture file **Format**.
2. Set the **Coloring**.
3. Specify the **File Type**, if applicable.
4. Define the **Resolution**, if applicable.
5. Set the appropriate **Options**.
6. If you are generating a window dump, specify the **Window Dump Command**.
7. (optional) Preview the result by clicking **Preview**.
8. Click the **Save...** button and enter the file name in the resulting **Select File** dialog box. See *Automatic Numbering of Files* (p. 61) for information on special features related to file name specification.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are not ready to save a picture but want to save the current picture settings, click the **Apply** button instead of the **Save...** button. The applied settings become the defaults for subsequent pictures.

4.22.1.1. Choosing the Picture File Format

To choose the picture file format, select one of the following items in the **Format** list:

EPS
(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

JPEG
is a common raster file format.

PPM
output is a common raster file format.

PostScript
is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF
is a common raster file format.

PNG
is a common raster file format.

VRML
is a graphics interchange format that allows export of 3D geometrical entities that you can display in the ANSYS FLUENT graphics window. This format can commonly be used by VR systems and the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in ANSYS FLUENT, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump
(Linux/UNIX systems only) selects a window dump operation for generating the picture. With this format, you need to specify the appropriate **Window Dump Command**.

4.22.1.2. Specifying the Color Mode

For all formats except VRML and the window dump, specify the type of **Coloring** you want to use for the picture file.

- Select **Color** for a color-scale copy.
- Select **Gray Scale** for a gray-scale copy.
- Select **Monochrome** for a black-and-white copy.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Most monochrome PostScript devices render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray Scale**.

4.22.1.3. Choosing the File Type

When you save an EPS (Encapsulated PostScript) or PostScript file, choose one of the following under **File Type**:

- A **Raster** file defines the color of each individual pixel in the image. Raster files have a fixed resolution. The supported raster formats are EPS, JPEG, PPM, PostScript, TIFF, and PNG.
- A **Vector** file defines the graphics image as a combination of geometric primitives like lines, polygons, and text. Vector files are usually scalable to any resolution. The supported vector formats include EPS, PostScript, and VRML.

Important

For the quickest print time, you can save vector files for simple 2D displays and raster files for complicated scenes.

4.22.1.4. Defining the Resolution

For raster picture files (i.e., JPEG, PPM, TIFF, and PNG), you can control the resolution of the picture image by specifying the size (in pixels). Set the desired **Width** and **Height** under **Resolution**. If the **Width** and **Height** are both zero, the picture is generated at the same resolution as the active graphics window. To check the size of the active window in pixels, click **Info** in the **Display Options** dialog box.

For EPS and PostScript files, specify the resolution in dots per inch (**DPI**) instead of setting the width and height.

4.22.1.5. Picture Options

For all picture formats except VRML and the window dump, you can control two additional settings under **Options**:

- Specify the orientation of the picture using the **Landscape Orientation** button. If this option is turned on, the picture is made in landscape mode; otherwise, it is made in portrait mode.
- Control the foreground/background color using the **White Background** option. If this option is enabled, the picture is saved with a white background, and (if you using the classic color scheme for the graphics window) the foreground color is changed to black.

4.22.2. Picture Options for PostScript Files

ANSYS FLUENT provides options that allow you to save PostScript files that can be printed more quickly. The following options are found in the `display/set/picture/driver/post-format` text menu:

`fast-raster`
enables a raster file that may be larger than the standard raster file, but will print much more quickly.

`raster`
enables the standard raster file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`rle-raster`
enables a run-length encoded raster file that is about the same size as the standard raster file, but will print slightly more quickly. This is the default file type.

`vector`
enables the standard vector file.

4.22.2.1. Window Dumps (Linux/UNIX Systems Only)

If you select the **Window Dump** format, the program uses the specified **Window Dump Command** to save the picture file. For example, if you want to use `xwd` to capture a window, set the **Window Dump Command** to

```
xwd -id %w >
```

When the dump occurs, ANSYS FLUENT automatically interprets `%w` to be the ID number of the active window.

When you click the **Save...** button, the **Select File** dialog box appears. Enter the file name for the output from the window dump (e.g., `myfile.xwd`).

If you are planning to make an animation, save the window dumps into numbered files, using the `%n` variable. To do this, use the **Window Dump Command** (`xwd -id %w`), but for the file name in the **Select File** dialog box enter `myfile%n.xwd`. Each time a new window dump is created, the value of `%n` increases by one. So there is no need to tack numbers onto the picture file names manually.

To use the **ImageMagick** animate program, saving the files in MIFF format (the native **ImageMagick** format) is more efficient. In such cases, use the **ImageMagick** tool `import`. Set the default **Window Dump Command** enter

```
import -window %w
```

Click **Save...** to invoke the **Select File** dialog box. Specify the output format to be MIFF by using the `.miff` suffix at the end of file name.

The window dump feature is both, system and graphics-driver specific. Thus the commands available for dumping windows depends on the particular configuration.

The window dump captures the window exactly as it is displayed, including resolution, colors, transparency, etc. For this reason, all of the inputs that control these characteristics are disabled in the **Save Picture** dialog box when you enable the **Window Dump** format. If you are using an 8-bit graphics display, use one of the built-in raster drivers (e.g., TIFF) to generate higher-quality 24-bit color output rather than dumping the 8-bit window.

4.22.2.2. Previewing the Picture Image

Before saving a picture file, you have the option of previewing what the saved image will look like. Click **Preview** to open a new window that will display the graphics using the current settings. This allows you to investigate the effects of different options interactively before saving the final, approved picture.

4.23. Setting Data File Quantities

By default, the information saved in a data file includes a standard set of quantities that were computed during the calculation, which are suitable for postprocessing in ANSYS FLUENT. If you plan to postprocess the data file in an application other than ANSYS FLUENT (e.g., ANSYS CFD-Post), however, you may want to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

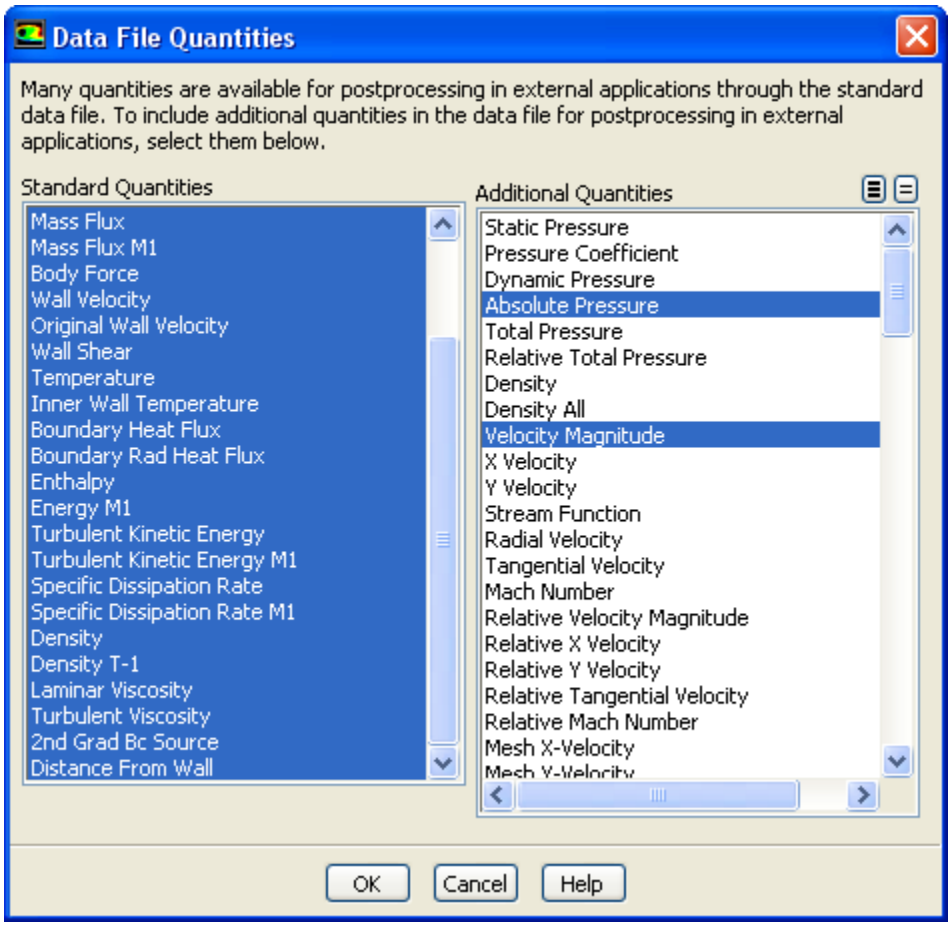
include additional quantities. The additional quantities that are available are derived from the standard quantities.

The procedure for generating a data file with additional quantities is as follows:

1. Set up the case file for your simulation.
2. Initialize the solution using the *Solution Initialization Task Page* (p. 1937).
3. Specify the quantities to be written in the data file, using the **Data File Quantities** dialog box (*Figure 4.19* (p. 122)).

   File → **Data File Quantities...**

   Figure 4.19 The Data File Quantities Dialog Box



   a. View the **Standard Quantities** list to see what will be saved in the data file by default. Note that you cannot deselect any of the **Standard Quantities**.
   b. Select the additional quantities you want saved from the **Additional Quantities** selection list.
   c. Click **OK**.
4. Save the case file. Note that the data file quantities you specified in the previous step will be saved as part of the case file.
5. Run the calculation and save the data file. This can be done as separate steps, or as one step if you have enabled the automatic saving of data files via the *Calculation Activities Task Page* (p. 1942).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Data File Quantities** dialog box can also be opened by clicking the **Data File Quantities...** button in the **Autosave Case/Data** dialog box.

4.24. The .fluent File

When starting up, ANSYS FLUENT looks in your home folder for an optional file called `.fluent`. If it finds the file, it loads it with the Scheme `load` function. This file can contain Scheme functions that customize the code's operation.

The `.fluent` file can also contain TUI commands that are executed via the Scheme function `ti-menu-load-string`. For example, if the `.fluent` file contains

```
(ti-menu-load-string "file read-case test.cas")
```

then ANSYS FLUENT will read in the case file `test.cas`. For more details about the function `ti-menu-load-string`, see *Text Menu Input from Character Strings* (p. 55).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Unit Systems

This chapter describes the units used in ANSYS FLUENT and how you can control them. Information is organized into the following sections:

5.1. Restrictions on Units
5.2. Units in Mesh Files
5.3. Built-In Unit Systems in ANSYS FLUENT
5.4. Customizing Units

ANSYS FLUENT allows you to work in any unit system, including inconsistent units. Thus, for example, you may work in British units with heat input in Watts or you may work in SI units with length defined in inches. This is accomplished by providing ANSYS FLUENT with a correct set of conversion factors between the units you want to use and the standard SI unit system that is used internally by the solver. ANSYS FLUENT uses these conversion factors for input and output, internally storing all parameters and performing all calculations in SI units. Both solvers always prompt you for the units required for all dimensional inputs.

Units can be altered part-way through a problem setup and/or after you have completed your calculation. If you have input some parameters in SI units and then you switch to British, all of your previous inputs (and the default prompts) are converted to the new unit system. If you have completed a simulation in SI units but you would like to report the results in any other units, you can alter the unit system and ANSYS FLUENT will convert all of the problem data to the new unit system when results are displayed. As noted above, all problem inputs and results are stored in SI units internally. This means that the parameters stored in the case and data files are in SI units. ANSYS FLUENT simply converts these values to your unit system at the interface level.

5.1. Restrictions on Units

It is important to note that the units for some inputs in ANSYS FLUENT are different from the units used for the rest of the problem setup.

- You must always define the following in SI units, regardless of the unit system you are using:
  - boundary profiles (see *Profiles* (p. 366))
  - source terms (see *Defining Mass, Momentum, Energy, and Other Sources* (p. 244))
  - custom field functions (see *Custom Field Functions* (p. 1579))
  - data in externally-created XY plot files (see *XY Plots of File Data* (p. 1472))
  - user-defined functions (See the UDF Manual for details about user-defined functions.)
- If you define a material property by specifying a temperature-dependent polynomial or piecewise-polynomial function, remember that temperature in the function is always in units of Kelvin or Rankine. If you are using Celsius or Kelvin as your temperature unit, then polynomial coefficient values must be entered in terms of Kelvin; if you are using Fahrenheit or Rankine as the temperature unit, values must be entered in terms of Rankine. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for information about temperature-dependent material properties.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.2. Units in Mesh Files

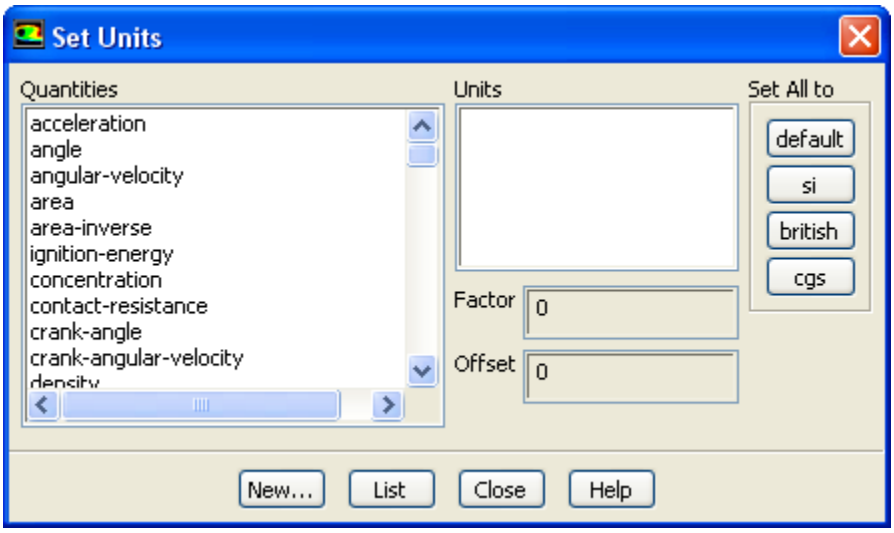
Some mesh generators allow you to define a set of units for the mesh dimensions. However, when you read the mesh into ANSYS FLUENT, it is always assumed that the unit of length is meters. If this is not true, you will need to scale the mesh, as described in *Scaling the Mesh* (p. 200).

5.3. Built-In Unit Systems in ANSYS FLUENT

ANSYS FLUENT provides four built-in unit systems: British, SI, CGS, and "default", all of which can be selected in the **Set Units** dialog box (*Figure 5.1* (p. 126)), using the buttons under the **Set All to** heading. To display the **Set Units** dialog box, select **General** in the navigation pane and click **Units...** in the task page.

General

Figure 5.1 The Set Units Dialog Box



To choose the English Engineering standard for all units, click the **british** button; to select the International System of units (SI) standard for all units, click the **si** button; to choose the CGS (centimeter-gram-second) standard for all units, click the **cgs** button; and to return to the "default" system, click the **default** button. The default system of units is similar to the SI system, but uses degrees instead of radians for angles. Clicking on one of the buttons under **Set All to** will immediately change the unit system. You can then close the dialog box if you are not interested in customizing any units.

Changing the unit system in the **Set Units** dialog box causes all future inputs that have units to be based on the newly selected unit system.

5.4. Customizing Units

If you would like a mixed unit system, or any unit system different from the four supplied by ANSYS FLUENT (and described in *Built-In Unit Systems in ANSYS FLUENT* (p. 126)), you can use the **Set Units** dialog box (*Figure 5.1* (p. 126)) to select an available unit or specify your own unit name and conversion factor for each quantity.

For additional information, please see the following sections:

5.4.1. Listing Current Units
5.4.2. Changing the Units for a Quantity

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4.3. Defining a New Unit

5.4.1. Listing Current Units

Before customizing units for one or more quantities, you may want to list the current units. You can do this by clicking the **List** button at the bottom of the **Set Units** dialog box. ANSYS FLUENT will print out a list (in the text window) containing all quantities and their current units, conversion factors, and offsets.

5.4.2. Changing the Units for a Quantity

ANSYS FLUENT will allow you to modify the units for individual quantities. This is useful for problems in which you want to use one of the built-in unit systems, but you want to change the units for one quantity (or for a few). For example, you may want to use SI units for your problem, but the dimensions of the geometry are given in inches. You can select the SI unit system, and then change the unit of length from meters to inches.

To change the units for a particular quantity, you will follow these two steps:

1. Select the quantity in the **Quantities** list (they are arranged in alphabetical order).
2. Choose a new unit from those that are available in the **Units** list.

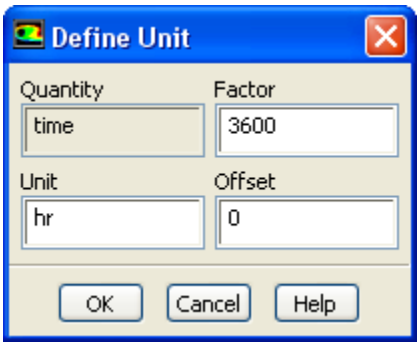
For the example cited above, you would choose **length** in the **Quantities** list, and then select **in** in the **Units** list. The **Factor** will automatically be updated to show 0.0254 meters/inch. (See *Figure 5.1* (p. 126).) If there was a non-zero offset for the new unit, the **Offset** field would also be updated. For example, if you were using SI units but wanted to define temperature in Celsius instead of Kelvin, you would select **temperature** in the **Quantities** list and **c** in the **Units** list. The **Factor** would change to 1, and the **Offset** would change to 273.15. Once you have selected the quantity and the new unit, no further action is needed, unless you wish to change the units for another quantity by following the same procedure.

5.4.3. Defining a New Unit

To create a new unit to be used for a particular quantity, you will follow the procedure below:

1. In the **Set Units** dialog box, select the quantity in the **Quantities** list.
2. Click the **New...** button and the **Define Unit** dialog box (*Figure 5.2* (p. 127)) will open. In this dialog box, the selected quantity will be shown in the **Quantity** field.

Figure 5.2 The Define Unit Dialog Box



3. Enter the name of your new unit in the **Unit** field, the conversion factor in the **Factor** field, and the offset in the **Offset** field.
4. Click **OK** in the **Define Unit** dialog box, and the new unit will appear in the **Set Units** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if you want to use hours as the unit of time, select **time** in the **Quantities** list in the **Set Units** dialog box and click the **New...** button. In the resulting **Define Unit** dialog box, enter `hr` for the **Unit** and `3600` for the **Factor**, as in *Figure 5.2* (p. 127). Then click **OK**. The new unit **hr** will appear in the **Units** list in the **Set Units** dialog box, and it will be selected.

5.4.3.1. Determining the Conversion Factor

The conversion factor you specify (**Factor** in the **Define Unit** dialog box) tells ANSYS FLUENT the number to multiply by to obtain the SI unit value from your customized unit value. Thus the conversion factor should have the form SI units/custom units. For example, if you want the unit of length to be inches, you should input a conversion factor of 0.0254 meters/inch. If you want the unit of velocity to be feet/min, you can determine the conversion factor by using the following equation:

$$x\frac{\mathrm{ft}}{\mathrm{min}}\times\frac{0.3048\mathrm{m}}{\mathrm{ft}}\times\frac{\mathrm{min}}{60\mathrm{s}}=y\frac{\mathrm{m}}{\mathrm{s}} \tag{5–1}$$

You should input a conversion factor of 0.0051, which is equal to 0.3048/60.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Reading and Manipulating Meshes

ANSYS FLUENT can import different types of meshes from various sources. You can modify the mesh by translating or scaling node coordinates, partitioning the cells for parallel processing, reordering the cells in the domain to decrease bandwidth, and merging or separating zones. You can convert all 3D meshes to polyhedral cells, except for pure hex meshes. Hexahedral cells are preserved during conversion. You can also obtain diagnostic information on the mesh, including memory usage and simplex, topological, and domain information. You can find out the number of nodes, faces, and cells in the mesh, determine the minimum and maximum cell volumes in the domain, and check for the proper numbers of nodes and faces per cell. These and other capabilities are described in the following sections.

6.1. Mesh Topologies
6.2. Mesh Requirements and Considerations
6.3. Mesh Import
6.4. Non-Conformal Meshes
6.5. Checking the Mesh
6.6. Reporting Mesh Statistics
6.7. Converting the Mesh to a Polyhedral Mesh
6.8. Modifying the Mesh

See *Adapting the Mesh* (p. 1325) for information about adapting the mesh based on solution data and related functions, and *Mesh Partitioning and Load Balancing* (p. 1603) for details on partitioning the mesh for parallel processing.

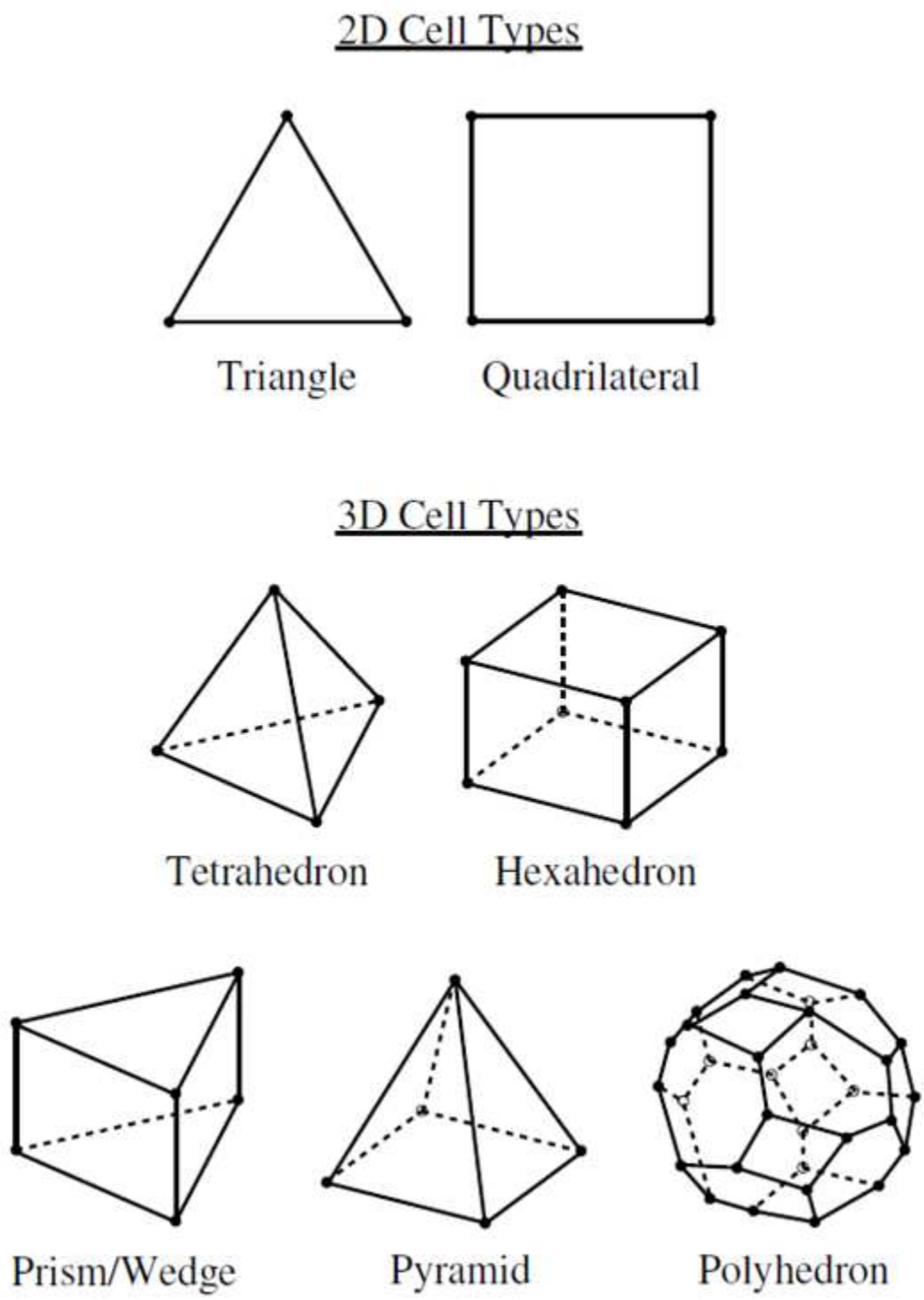
6.1. Mesh Topologies

As an unstructured solver, ANSYS FLUENT uses internal data structures to assign an order to the cells, faces, and grid points in a mesh and to maintain contact between adjacent cells. Therefore, it does not require i,j,k indexing to locate neighboring cells. This gives you the flexibility to use the best mesh topology for your problem, as the solver does not force an overall structure or topology on the mesh.

For 2D meshes, quadrilateral and triangular cells are accepted, and for 3D meshes, hexahedral, tetrahedral, pyramid, wedge, and polyhedral cells can be used. *Figure 6.1* (p. 130) depicts each of these cell types. Both single-block and multi-block structured meshes, as well as hybrid meshes containing quadrilateral and triangular cells or hexahedral, tetrahedral, pyramid, and wedge cells are acceptable. ANSYS FLUENT also accepts meshes with hanging nodes (i.e., nodes on edges and faces that are not vertices of all the cells sharing those edges or faces), although you may need to remove hanging nodes from interior walls. See Hanging Node Adaption in the Theory Guide for information about acceptable hanging nodes, and *Using the tpoly Filter to Remove Hanging Nodes* (p. 155) for details on removing hanging nodes. Meshes with non-conformal boundaries (i.e., meshes with multiple subdomains in which the mesh node locations at the internal subdomain boundaries are not identical) are also acceptable. For details, see *Non-Conformal Meshes* (p. 161).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.1 Cell Types



Some examples of meshes that are valid for ANSYS FLUENT are presented in *Examples of Acceptable Mesh Topologies* (p. 131). Different cell shapes and their face-node connectivity are explained in *Face-Node Connectivity in ANSYS FLUENT* (p. 136). *Choosing the Appropriate Mesh Type* (p. 141) explains how to choose the mesh type that is best suited for your problem.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.1. Examples of Acceptable Mesh Topologies

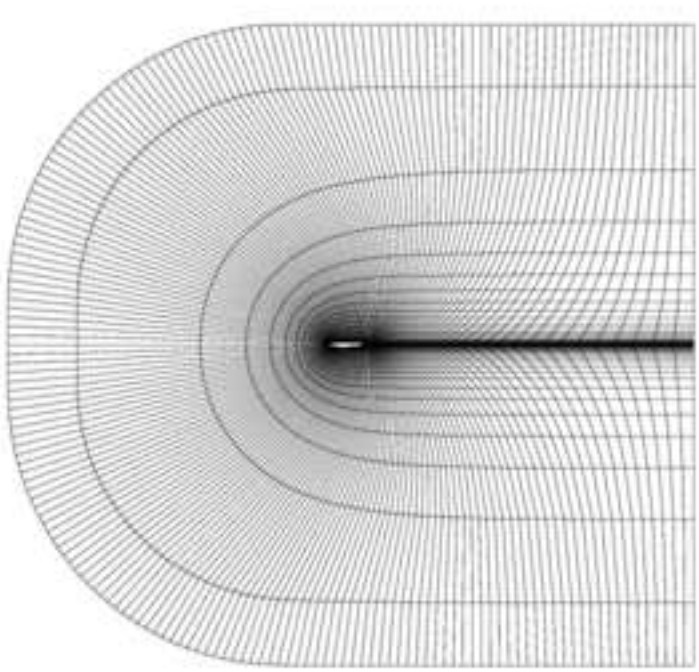
ANSYS FLUENT can solve problems on a wide variety of meshes. *Figure 6.2* (p. 131)-*Figure 6.13* (p. 136) show examples of meshes that are valid for ANSYS FLUENT.

O-type meshes, meshes with zero-thickness walls, C-type meshes, conformal block-structured meshes, multiblock structured meshes, non-conformal meshes, and unstructured triangular, tetrahedral, quadrilateral, hexahedral, and polyhedral meshes are all acceptable.

Note

Though ANSYS FLUENT does not require a cyclic branch cut in an O-type mesh, it will accept a mesh that contains one.

Figure 6.2 Structured Quadrilateral Mesh for an Airfoil



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.3 Unstructured Quadrilateral Mesh

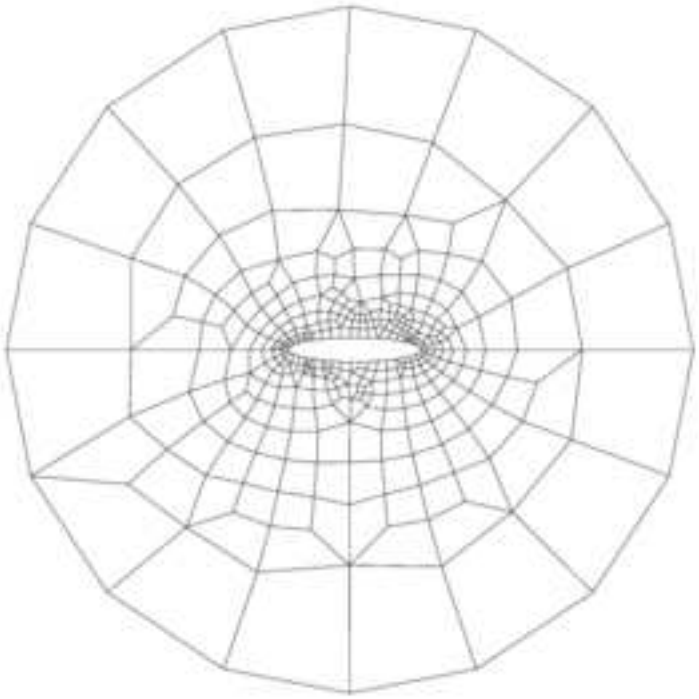


Figure 6.4 Multiblock Structured Quadrilateral Mesh

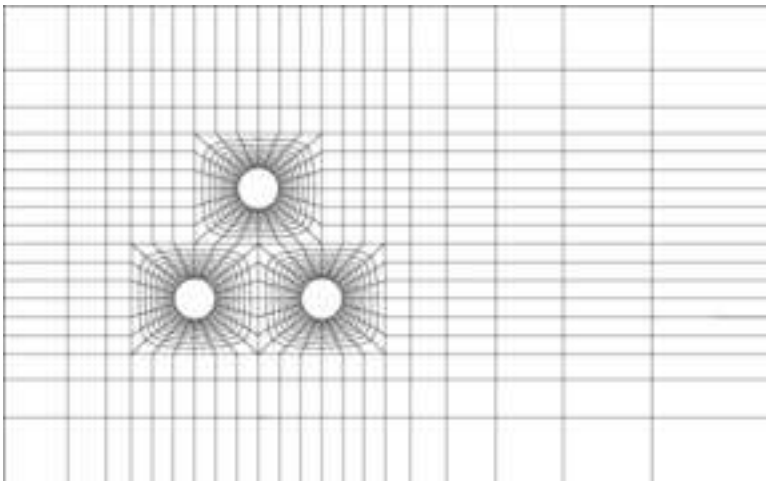
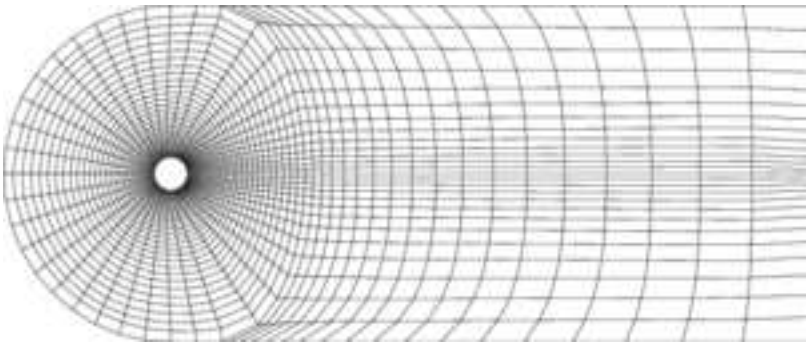


Figure 6.5 O-Type Structured Quadrilateral Mesh



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.6 Parachute Modeled With Zero-Thickness Wall

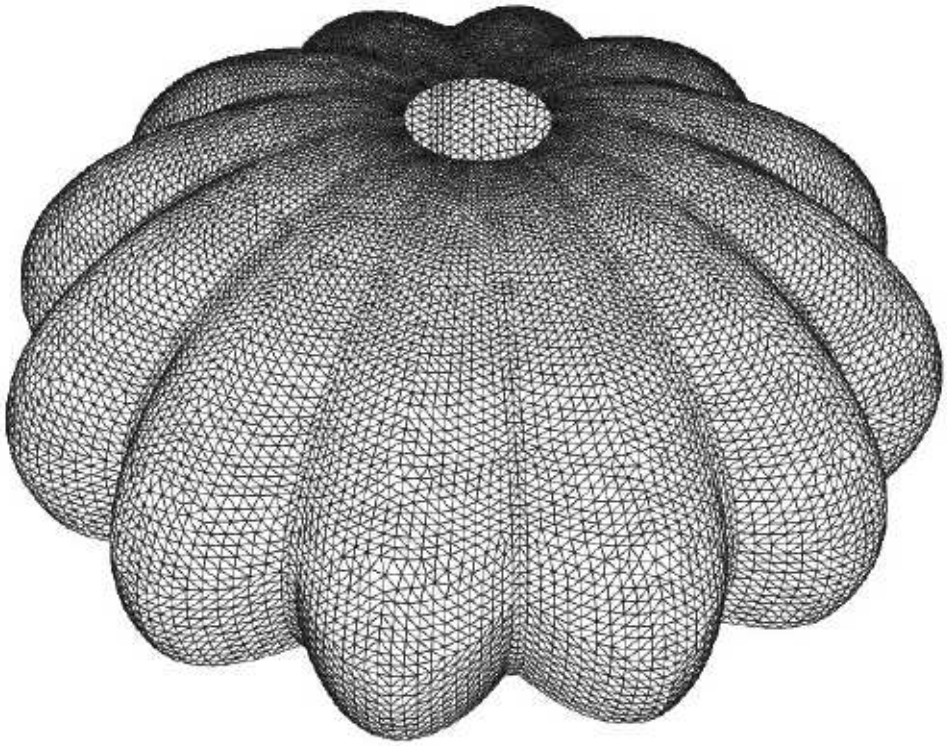
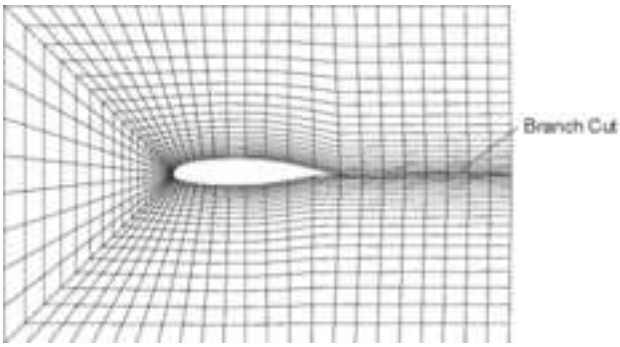


Figure 6.7 C-Type Structured Quadrilateral Mesh



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.8 3D Multiblock Structured Mesh

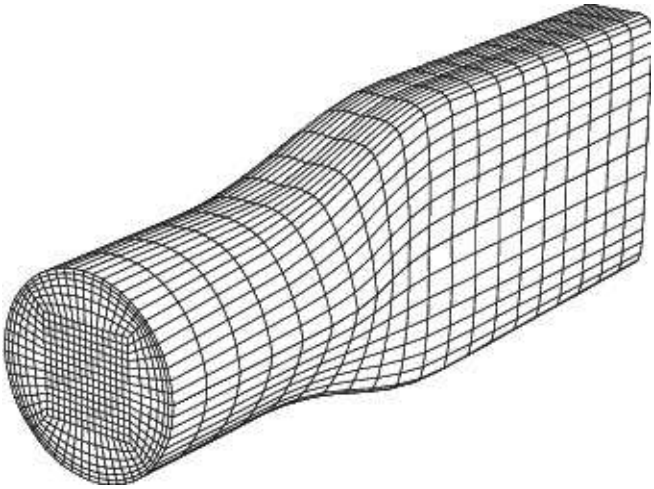
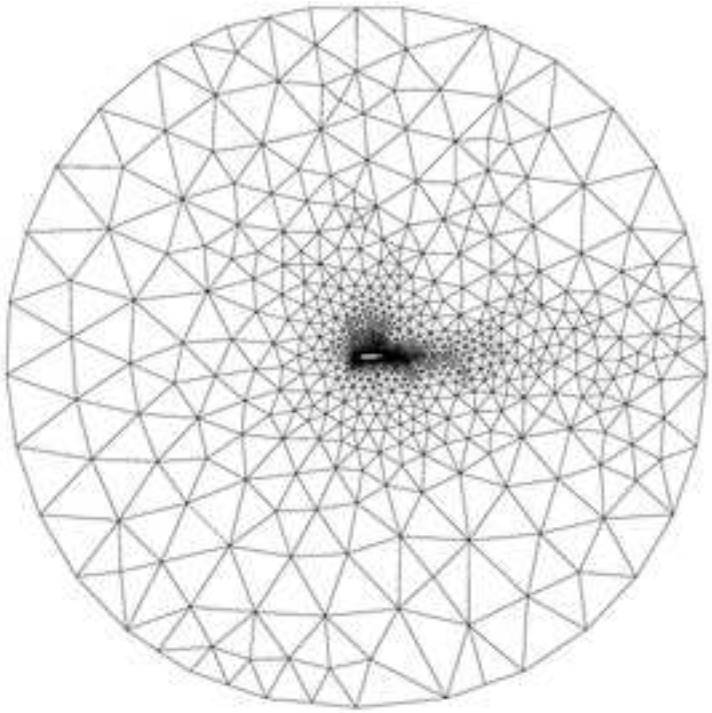


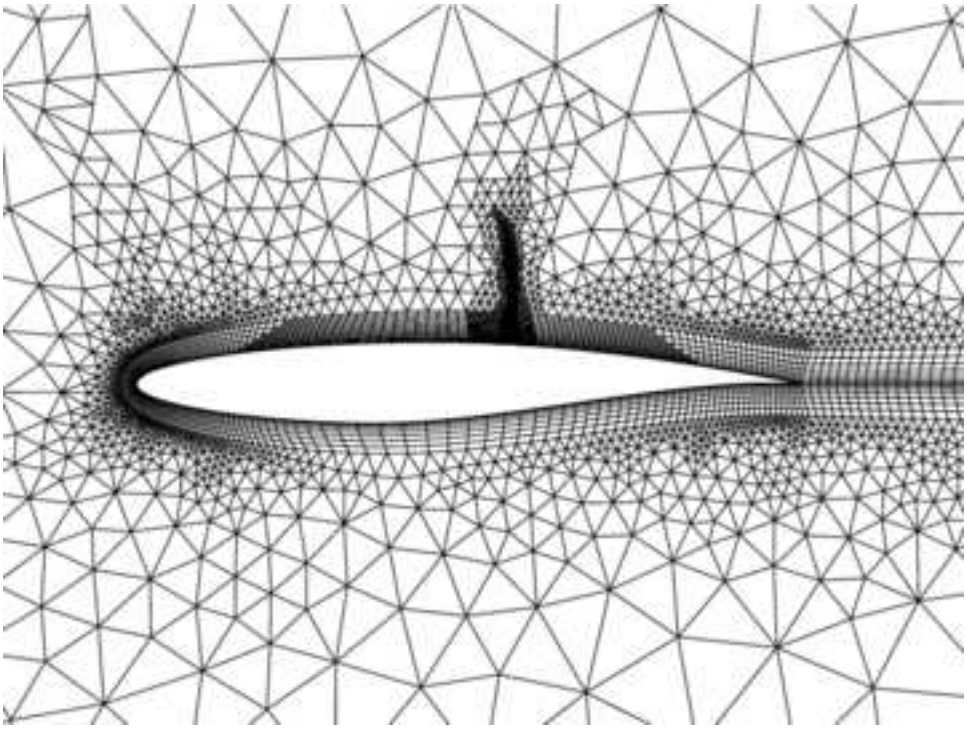
Figure 6.9 Unstructured Triangular Mesh for an Airfoil



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.10 Unstructured Tetrahedral Mesh

Figure 6.11 Hybrid Triangular/Quadrilateral Mesh with Hanging Nodes



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.12 Non-Conformal Hybrid Mesh for a Rotor-Stator Geometry

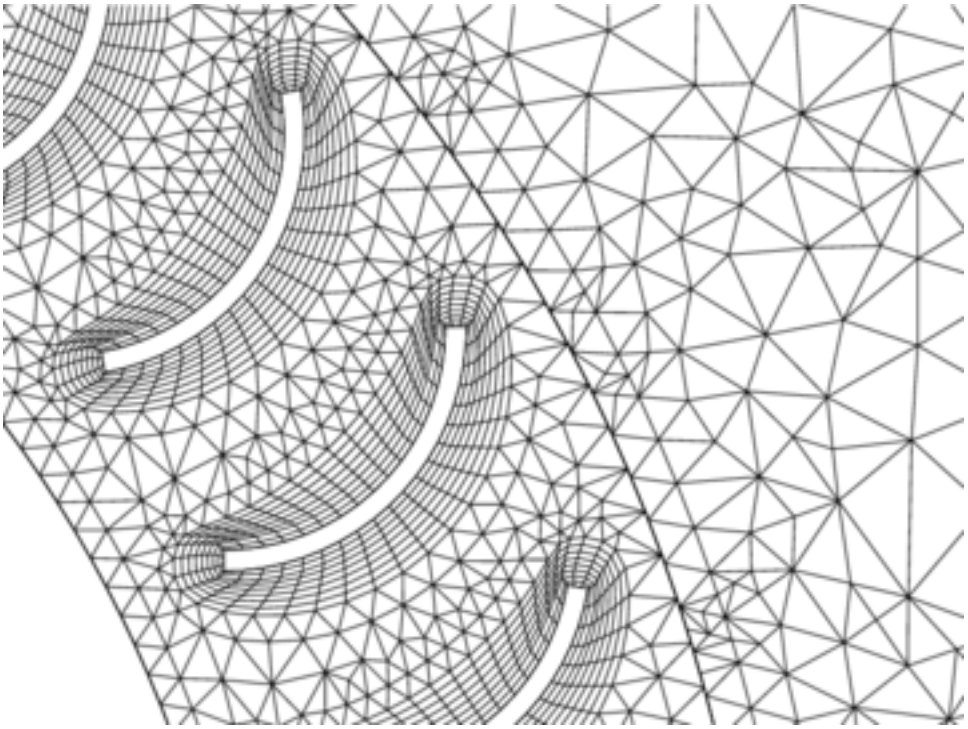


Figure 6.13 Polyhedral Mesh

6.1.2. Face-Node Connectivity in ANSYS FLUENT

This section contains information about the connectivity of faces and their related nodes in terms of node number and face number.

Face-node connectivity for the following cell shapes is explained here:

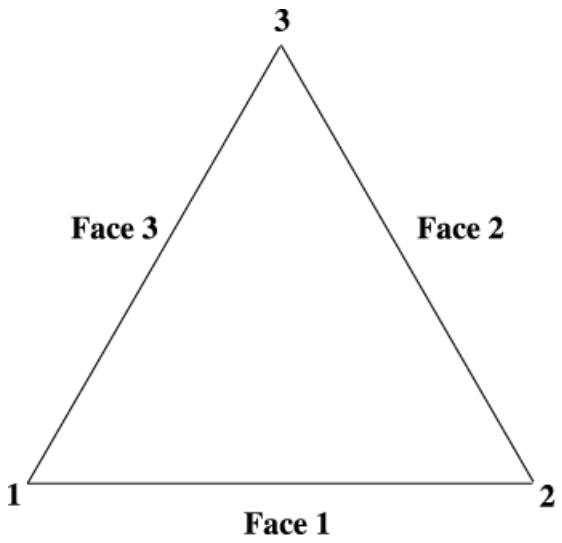
- triangular (*Figure 6.14* (p. 137))
- quadrilateral (*Figure 6.15* (p. 137))
- tetrahedral (*Figure 6.16* (p. 138))
- wedge (*Figure 6.17* (p. 139))
- pyramidal (*Figure 6.18* (p. 140))
- hex (*Figure 6.19* (p. 140))
- polyhedral (*Figure 6.20* (p. 141))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This information is useful in interfacing with ANSYS FLUENT.

6.1.2.1. Face-Node Connectivity for Triangular Cells

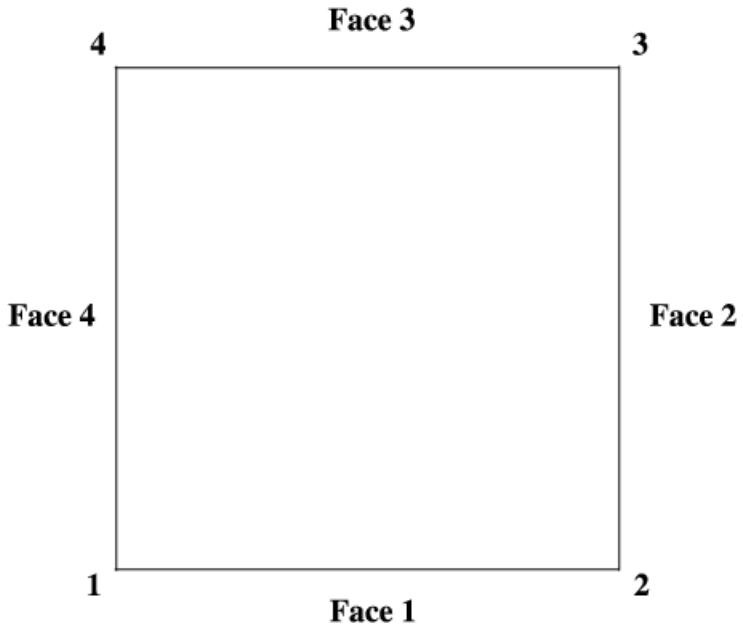
Figure 6.14 Face and Node Numbering for Triangular Cells



Face	Associated Nodes
Face 1	1-2
Face 2	2-3
Face 3	3-1

6.1.2.2. Face-Node Connectivity for Quadrilateral Cells

Figure 6.15 Face and Node Numbering for Quadrilateral Cells

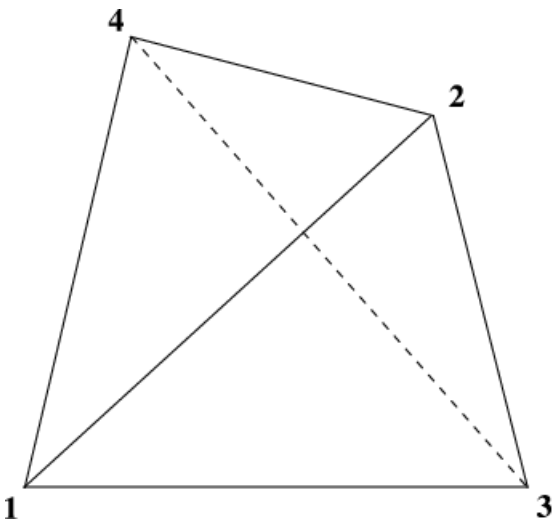


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Face	Associated Nodes
Face 1	1-2
Face 2	2-3
Face 3	3-4
Face 4	4-1

6.1.2.3. Face-Node Connectivity for Tetrahedral Cells

Figure 6.16 Face and Node Numbering for Tetrahedral Cells

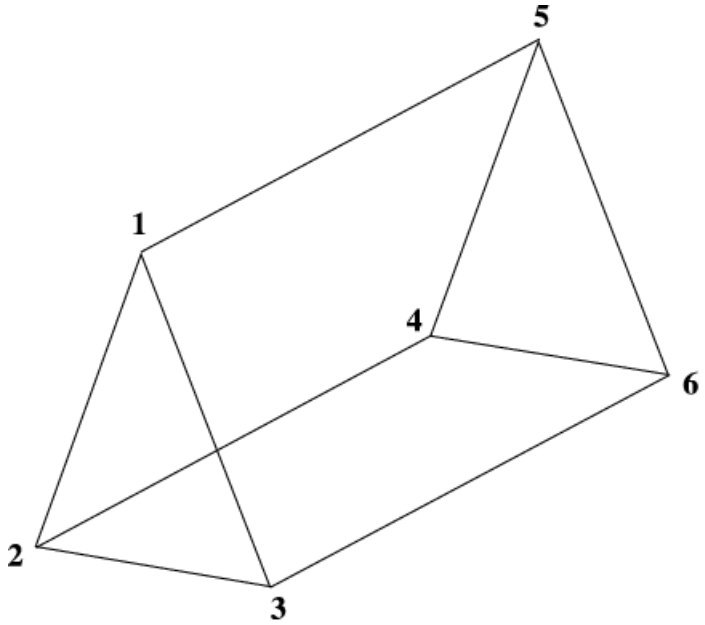


Face	Associated Nodes
Face 1	3-2-4
Face 2	4-1-3
Face 3	2-1-4
Face 4	3-1-2

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.4. Face-Node Connectivity for Wedge Cells

Figure 6.17 Face and Node Numbering for Wedge Cells

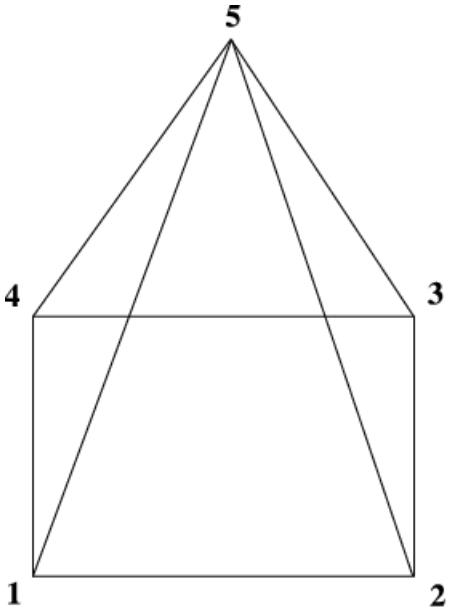


Face	Associated Nodes
Face 1	3-2-1
Face 2	6-5-4
Face 3	4-2-3-6
Face 4	5-1-2-4
Face 5	6-3-1-5

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.5. Face-Node Connectivity for Pyramidal Cells

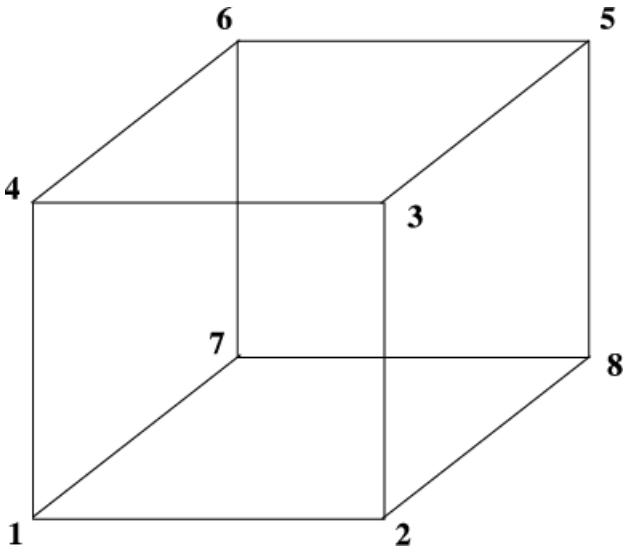
Figure 6.18 Face and Node Numbering for Pyramidal Cells



Face	Associated Nodes
Face 1	4-3-2-1
Face 2	4-5-3
Face 3	3-5-2
Face 4	2-5-1
Face 5	1-5-4

6.1.2.6. Face-Node Connectivity for Hex Cells

Figure 6.19 Face and Node Numbering for Hex Cells

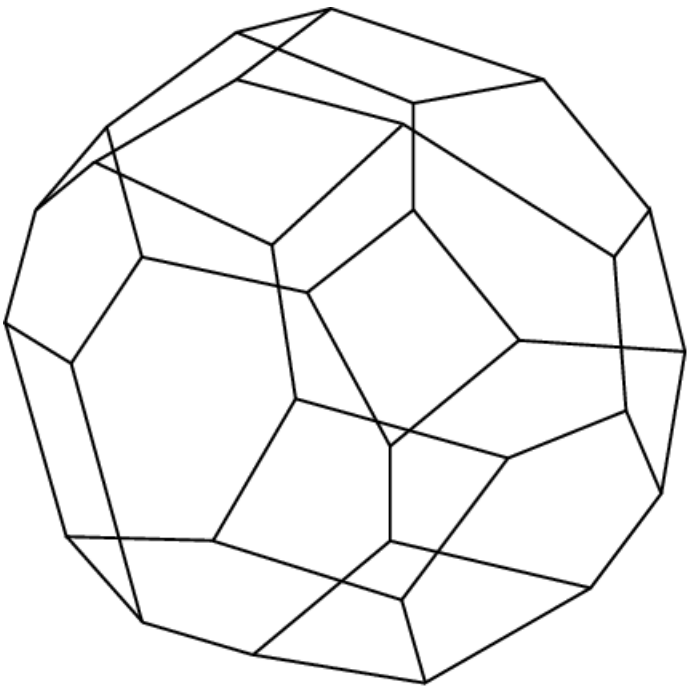


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Face	Associated Nodes
Face 1	4-3-2-1
Face 2	3-4-6-5
Face 3	4-1-7-6
Face 4	2-3-5-8
Face 5	1-2-8-7
Face 6	7-8-5-6

6.1.2.7. Face-Node Connectivity for Polyhedral Cells

Figure 6.20 An Example of a Polyhedral Cell



For polyhedral cells, there is no explicit face and node numbering as with the other cell types.

6.1.3. Choosing the Appropriate Mesh Type

ANSYS FLUENT can use meshes comprised of triangular or quadrilateral cells (or a combination of the two) in 2D, and tetrahedral, hexahedral, polyhedral, pyramid, or wedge cells (or a combination of these) in 3D. The choice of which mesh type to use will depend on your application. When choosing mesh type, consider the following issues:

- setup time
- computational expense
- numerical diffusion

6.1.3.1. Setup Time

Many flow problems solved in engineering practice involve complex geometries. The creation of structured or block-structured meshes (consisting of quadrilateral or hexahedral elements) for such problems can be

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

extremely time-consuming if not impossible. Therefore, setup time for complex geometries is the major motivation for using unstructured meshes employing triangular or tetrahedral cells. However, if your geometry is relatively simple, there may be no saving in setup time with either approach.

Other risks of using structured or block-structured meshes with complicated geometries include the over-simplification of the geometry, mesh quality issues, and a less efficient mesh distribution (e.g., fine resolution in areas of less importance) that results in a high cell count.

If you already have a mesh created for a structured code, it will save you time to use this mesh in ANSYS FLUENT rather than regenerate it. This can be a motivation for using quadrilateral or hexahedral cells in your ANSYS FLUENT simulation.

Note

ANSYS FLUENT has a range of filters that allow you to import structured meshes from other codes, including FLUENT 4. For details, see *Mesh Import* (p. 148).

6.1.3.2. Computational Expense

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows clustering of cells in selected regions of the flow domain. Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

A characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. Therefore, if you have a relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, use a mesh of high-aspect-ratio quadrilateral/hexahedral cells. The mesh is likely to have far fewer cells than if you use triangular/tetrahedral cells.

Converting the entire domain of your (tetrahedral) mesh to a polyhedral mesh will result in a lower cell count than your original mesh. Although the result is a coarser mesh, convergence will generally be faster, possibly saving you some computational expense.

In summary, the following practices are generally recommended:

- For simple geometries, use quadrilateral/hexahedral meshes.
- For moderately complex geometries, use unstructured quadrilateral/hexahedral meshes.
- For relatively complex geometries, use triangular/tetrahedral meshes with prism layers.
- For extremely complex geometries, use pure triangular/tetrahedral meshes.

6.1.3.3. Numerical Diffusion

A dominant source of error in multidimensional situations is numerical diffusion (false diffusion). The term *false diffusion* is used because the diffusion is not a real phenomenon, yet its effect on a flow calculation is analogous to that of increasing the real diffusion coefficient.

The following comments can be made about numerical diffusion:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Numerical diffusion is most noticeable when the real diffusion is small, that is, when the situation is convection-dominated.
- All practical numerical schemes for solving fluid flow contain a finite amount of numerical diffusion. This is because numerical diffusion arises from truncation errors that are a consequence of representing the fluid flow equations in discrete form.
- The second-order and the MUSCL discretization scheme used in ANSYS FLUENT can help reduce the effects of numerical diffusion on the solution.
- The amount of numerical diffusion is inversely related to the resolution of the mesh. Therefore, one way of dealing with numerical diffusion is to refine the mesh.
- Numerical diffusion is minimized when the flow is aligned with the mesh.

  This is the most relevant to the choice of the mesh. If you use a triangular/tetrahedral mesh, the flow can *never* be aligned with the mesh. If you use a quadrilateral/hexahedral mesh, this situation might occur, but *not for complex flows*. It is only in a *simple* flow, such as the flow through a long duct, in which you can rely on a quadrilateral/hexahedral mesh to minimize numerical diffusion. In such situations, it is advantageous to use a quadrilateral/hexahedral mesh, since you will be able to get a better solution with fewer cells than if you were using a triangular/tetrahedral mesh.
- If you would like higher resolution for a gradient that is perpendicular to a wall, you can create prism layers with higher aspect ratios near the wall.

6.2. Mesh Requirements and Considerations

This section contains information about special geometry/mesh requirements and general comments on mesh quality.

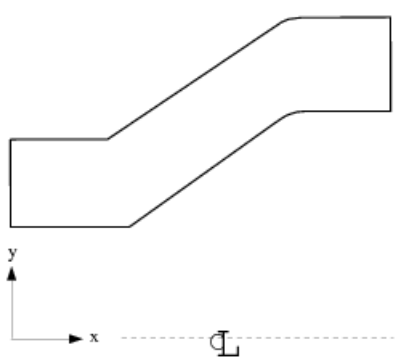
6.2.1. Geometry/Mesh Requirements
6.2.2. Mesh Quality

6.2.1. Geometry/Mesh Requirements

You should be aware of the following geometry setup and mesh construction requirements at the beginning of your problem setup:

- Axisymmetric geometries must be defined such that the axis of rotation is the x axis of the Cartesian coordinates used to define the geometry (*Figure 6.21* (p. 143)).

  Figure 6.21 Setup of Axisymmetric Geometries with the x Axis as the Centerline



- ANSYS FLUENT allows you to set up periodic boundaries using either conformal or non-conformal periodic zones. For conformal periodic boundaries, the periodic zones must have identical meshes.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The conformal periodic boundaries can be created in GAMBIT or TGrid when you are generating the volume mesh. See the GAMBIT Modeling Guide or the TGrid User's Guide for more information about creating periodic boundaries in GAMBIT or TGrid.) Alternatively, you can create the conformal periodic boundaries in ANSYS FLUENT using the `mesh/modify-zones/make-periodic` text command. For details, see *Creating Conformal Periodic Zones* (p. 191).

Although GAMBIT and TGrid can produce true periodic boundaries, most CAD packages do not. If your mesh was created in such a package, set up a non-conformal interface with the periodic boundary condition option enabled in ANSYS FLUENT. For details, see *Creating Conformal Periodic Zones* (p. 191).

6.2.2. Mesh Quality

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Regardless of the type of mesh used in your domain, checking the quality of your mesh is essential. One important indicator of mesh quality that ANSYS FLUENT allows you to check is a quantity referred to as the orthogonal quality. In order to determine the orthogonal quality of a given cell, the following quantities are calculated for each face i:

- the normalized dot product of the area vector of a face ($\vec{A}_i$) and a vector from the centroid of the cell to the centroid of that face ($\vec{f}_i$):

$$\frac{\vec{A}_i \cdot \vec{f}_i}{\left|\vec{A}_i\right|\left|\vec{f}_i\right|} \tag{6–1}$$

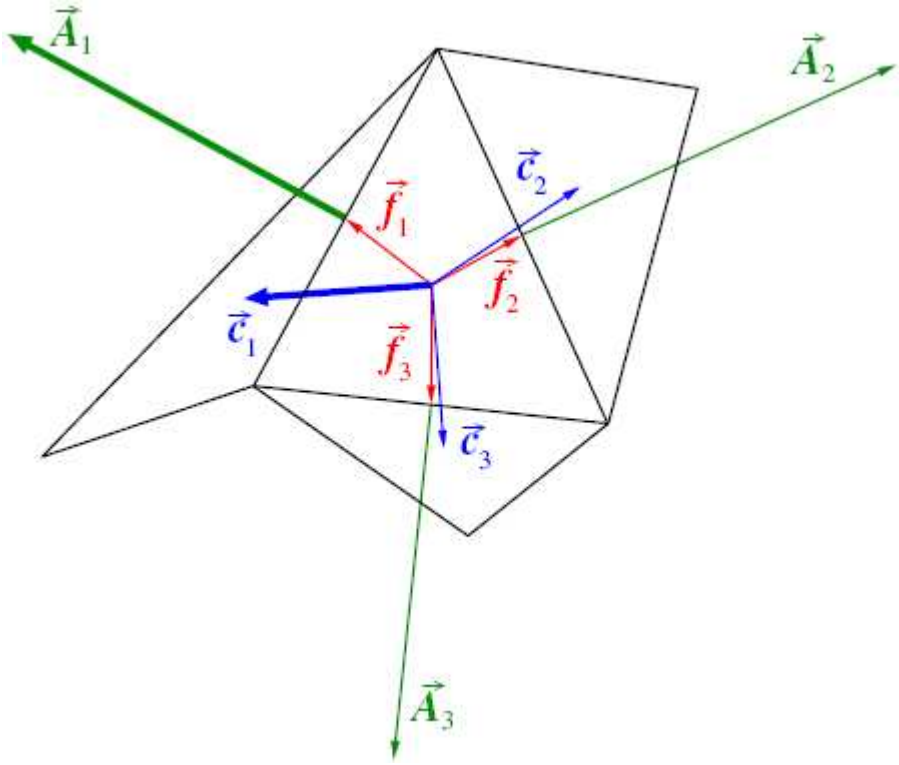
- the normalized dot product of the area vector of a face ($\vec{A}_i$) and a vector from the centroid of the cell to the centroid of the adjacent cell that shares that face ($\vec{c}_i$):

$$\frac{\vec{A}_i \cdot \vec{c}_i}{\left|\vec{A}_i\right|\left|\vec{c}_i\right|} \tag{6–2}$$

The minimum value that results from calculating *Equation 6–1* (p. 144)and *Equation 6–2* (p. 144) for all of the faces is then defined as the orthogonal quality for the cell. Therefore, the worst cells will have an orthogonal quality closer to 0 and the best cells will have an orthogonal quality closer to 1. *Figure 6.22* (p. 145) illustrates the relevant vectors, and is an example where *Equation 6–2* (p. 144) produces the minimum value and therefore determines the orthogonal quality.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

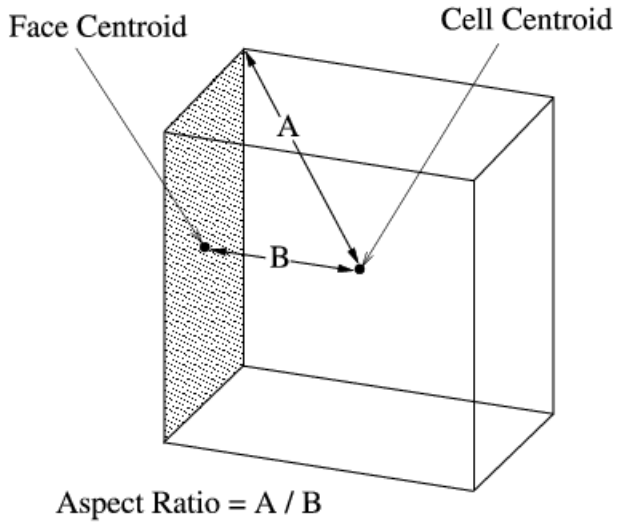
Figure 6.22 The Vectors Used to Compute Orthogonal Quality



Another important indicator of the mesh quality is the aspect ratio. The aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the distances between the cell centroid and face centroids, and the distances between the cell centroid and nodes. For a unit cube (see *Figure 6.23* (p. 146)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.23 Calculating the Aspect Ratio for a Unit Cube



To check the quality of your mesh, you can use the **Report Quality** button in the **General** task page:

General → **Report Quality**

A message will be displayed in the console, such as the example that follows:

```
Mesh Quality:
 Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality = 6.07960e-01
Maximum Aspect Ratio = 5.42664e+00
```

If you would like more information about the quality displayed in the console (including additional quality metrics and the zones that have the cells with the lowest quality), set the `mesh/check-verbosity` text command to 2 prior to using the **Report Quality** button.

For information about how to improve poor quality cells, see *Repairing Meshes* (p. 174).

When evaluating whether the quality of your mesh is sufficient for the problem you are modeling, it is important to consider attributes such as mesh element distribution, cell shape, smoothness, and flow-field dependency. These attributes are described in the sections that follow.

6.2.2.1. Mesh Element Distribution

Since you are discretely defining a continuous domain, the degree to which the salient features of the flow (such as shear layers, separated regions, shock waves, boundary layers, and mixing zones) are resolved depends on the density and distribution of mesh elements. In many cases, poor resolution in critical regions can dramatically affect results. For example, the prediction of separation due to an adverse pressure gradient depends heavily on the resolution of the boundary layer upstream of the point of separation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resolution of the boundary layer (i.e., mesh spacing near walls) also plays a significant role in the accuracy of the computed wall shear stress and heat transfer coefficient. This is particularly true in laminar flows where the mesh adjacent to the wall should obey

$$y_p\sqrt{\frac{u_\infty}{\nu x}} \leq 1 \qquad \textbf{(6–3)}$$

where y_p = distance to the wall from the adjacent cell centroid

u_∞ = free-stream velocity

ν = kinematic viscosity of the fluid

x = distance along the wall from the starting point of the boundary layer

Equation 6–3 (p. 147) is based upon the Blasius solution for laminar flow over a flat plate at zero incidence [*75*] (p. 2213).

Proper resolution of the mesh for turbulent flows is also very important. Due to the strong interaction of the mean flow and turbulence, the numerical results for turbulent flows tend to be more susceptible to mesh element distribution than those for laminar flows. In the near-wall region, different mesh resolutions are required depending on the near-wall model being used. See *Model Hierarchy* (p. 654) for guidelines.

In general, no flow passage should be represented by fewer than 5 cells. Most cases will require many more cells to adequately resolve the passage. In regions of large gradients, as in shear layers or mixing zones, the mesh should be fine enough to minimize the change in the flow variables from cell to cell. Unfortunately, it is very difficult to determine the locations of important flow features in advance. Moreover, the mesh resolution in most complicated 3D flow fields will be constrained by CPU time and computer resource limitations (i.e., memory and disk space). Although accuracy increases with larger meshes, the CPU and memory requirements to compute the solution and postprocess the results also increase. Solution-adaptive mesh refinement can be used to increase and/or decrease mesh density based on the evolving flow field, and thus provides the potential for more economical use of grid points (and hence reduced time and resource requirements). See *Adapting the Mesh* (p. 1325) for information on solution adaption.

6.2.2.2. Cell Quality

The quality of the cell (including its orthogonal quality, aspect ratio, and skewness) also has a significant impact on the accuracy of the numerical solution.

- *Orthogonal quality* is computed for cells using the vector from the cell centroid to each of its faces, the corresponding face area vector, and the vector from the cell centroid to the centroids of each of the adjacent cells (see *Equation 6–1* (p. 144), *Equation 6–2* (p. 144), and *Figure 6.22* (p. 145)). The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher.
- *Aspect ratio* is a measure of the stretching of the cell. As discussed in *Computational Expense* (p. 142), for highly anisotropic flows, extreme aspect ratios may yield accurate results with fewer cells. Generally, it is best to avoid sudden and large changes in cell aspect ratios in areas where the flow field exhibit large changes or strong gradients.
- *Skewness* is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees. A general rule is that the maximum skewness for a triangular/tetrahedral mesh in most flows should be kept below

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.95, with an average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties and may require changing the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.

Cell size change and face warp are additional quality measure that could affect stability and accuracy. See the TGrid User's Guide for more details.

6.2.2.3. Smoothness

Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations. Rapid changes in cell volume between adjacent cells translate into larger truncation errors. ANSYS FLUENT provides the capability to improve the smoothness by refining the mesh based on the change in cell volume or the gradient of cell volume. For information on refining the mesh based on change in cell volume, see *Gradient Adaption* (p. 1331) and *Volume Adaption* (p. 1337).

6.2.2.4. Flow-Field Dependency

The effect of resolution, smoothness, and cell shape on the accuracy and stability of the solution process is dependent on the flow field being simulated. For example, very skewed cells can be tolerated in benign flow regions, but can be very damaging in regions with strong flow gradients.

Since the locations of strong flow gradients generally cannot be determined a priori, you should strive to achieve a high-quality mesh over the entire flow domain.

6.3. Mesh Import

Since ANSYS FLUENT can handle a number of different mesh topologies, there are many sources from which you can obtain a mesh to be used in your simulation. You can generate a mesh using GAMBIT, TGrid, GeoMesh, PreBFC, ICEM CFD, I-deas, NASTRAN, PATRAN, ARIES, Mechanical APDL, ANSYS CFX, or other preprocessors. You can also use the mesh contained in a FLUENT/UNS, RAMPANT, or FLUENT 4 case file. You can also prepare multiple mesh files and combine them to create a single mesh.

6.3.1. GAMBIT Mesh Files
6.3.2. GeoMesh Mesh Files
6.3.3. TGrid Mesh Files
6.3.4. PreBFC Mesh Files
6.3.5. ICEM CFD Mesh Files
6.3.6. I-deas Universal Files
6.3.7. NASTRAN Files
6.3.8. PATRAN Neutral Files
6.3.9. Mechanical APDL Files
6.3.10. ANSYS CFX Files
6.3.11. Using the fe2ram Filter to Convert Files
6.3.12. Using the tpoly Filter to Remove Hanging Nodes
6.3.13. FLUENT/UNS and RAMPANT Case Files
6.3.14. FLUENT 4 Case Files
6.3.15. ANSYS FIDAP Neutral Files
6.3.16. Reading Multiple Mesh/Case/Data Files
6.3.17. Reading Surface Mesh Files

6.3.1. GAMBIT Mesh Files

You can use GAMBIT to create 2D and 3D structured/unstructured/hybrid meshes. To create any of these meshes for ANSYS FLUENT, follow the procedure described in the GAMBIT Modeling Guide, and export your

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh in ANSYS FLUENT 5/6 format. All such meshes can be imported directly into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.2. GeoMesh Mesh Files

You can use GeoMesh to create complete 2D quadrilateral or triangular meshes, 3D hexahedral meshes, and triangular surface meshes for 3D tetrahedral meshes. To create any of these meshes for ANSYS FLUENT, follow the procedure described in the GeoMesh User's Guide.

To complete the generation of a 3D tetrahedral mesh, read the surface mesh into TGrid and generate the volume mesh there. All other meshes can be imported directly into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.3. TGrid Mesh Files

You can use TGrid to create 2D and 3D unstructured triangular/tetrahedral meshes from boundary or surface meshes. Follow the meshing procedure described in the TGrid User's Guide, and save your mesh using the **File/Write/Mesh...** menu item. To import the mesh into ANSYS FLUENT, use the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.4. PreBFC Mesh Files

You can use PreBFC to create two different types of meshes for ANSYS FLUENT, structured quadrilateral/hexahedral and unstructured triangular/tetrahedral.

6.3.4.1. Structured Mesh Files

To generate a structured 2D or 3D mesh, follow the procedure described in the PreBFC User's Guide (Chapters 6 and 7). The resulting mesh will contain quadrilateral (2D) or hexahedral (3D) elements. Do not specify more than 70 wall zones and 35 inlet zones.

To import the mesh, use the **File/Import/PreBFC File...** menu item, as described in *PreBFC Files* (p. 83).

To manually convert a file in PreBFC format to a mesh file suitable for ANSYS FLUENT, enter the following command:

`utility fl42seg` *input_filename output_filename*

The output file produced can be read into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.4.2. Unstructured Triangular and Tetrahedral Mesh Files

To generate an unstructured 2D mesh, follow the procedure described in the PreBFC User's Guide. Save the mesh file in the RAMPANT format using the `MESH-RAMPANT/TGRID` command. The current ANSYS FLUENT format is the same as the RAMPANT format. The resulting mesh will contain triangular elements. To import the mesh, use the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

To generate a 3D unstructured tetrahedral mesh, follow the procedure described in Chapter 8 of the PreBFC User's Guide for generating a surface mesh. Then read the surface mesh into TGrid, and complete the mesh generation there. See *TGrid Mesh Files* (p. 149) for information about TGrid mesh files.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.5. ICEM CFD Mesh Files

You can use ICEM CFD to create structured meshes in FLUENT 4 format and unstructured meshes in RAMPANT format.

- To import a FLUENT 4 mesh, follow the instructions in *FLUENT 4 Case Files* (p. 156).
- To import a RAMPANT mesh, use the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

The current ANSYS FLUENT format is the same as the RAMPANT format,*not* the FLUENT 4 format. After reading a triangular or tetrahedral ICEM CFD volume mesh, perform smoothing and swapping (as described in *Improving the Mesh by Smoothing and Swapping* (p. 1349)) to improve its quality.

6.3.6. I-deas Universal Files

You can import an I-deas Universal file into ANSYS FLUENT in three different ways.

- Generate an I-deas surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge and/or hexahedral elements. Import it into TGrid using the commands described in the TGrid User's Guide. Adhere to the restrictions described in Appendix B of the TGrid User's Guide. In TGrid, complete the mesh generation (if necessary) and follow the instructions in *TGrid Mesh Files* (p. 149).
- Generate an I-deas volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Import it directly using the **File/Import/I-deas Universal...** menu item, as described in *I-deas Universal Files* (p. 80).
- Generate an I-deas volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Use the `fe2ram` filter to convert the Universal file to the format used by ANSYS FLUENT. To convert an input file in I-deas Universal format to an output file in ANSYS FLUENT format, follow the instructions below in *Using the fe2ram Filter to Convert Files* (p. 155). After the output file is written, read it into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.6.1. Recognized I-deas Datasets

The following Universal file datasets are recognized by the ANSYS FLUENT mesh import utility:

- **Node Coordinates** dataset number 15, 781, 2411
- **Elements** dataset number 71, 780, 2412
- **Permanent Groups** dataset number 752, 2417, 2429, 2430, 2432, 2435

For 2D volume meshes, the elements must exist in a constant z plane.

Note

The mesh area or mesh volume datasets are *not* recognized. This implies that writing multiple mesh areas/volumes to a single Universal file may confuse ANSYS FLUENT.

6.3.6.2. Grouping Nodes to Create Face Zones

Nodes are grouped in I-deas using the `Group` command to create boundary face zones. In ANSYS FLUENT, boundary conditions are applied to each zone. Faces that contain the nodes in a group are gathered into a single zone. It is important not to group nodes of internal faces with nodes of boundary faces.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One technique is to generate groups automatically based on curves or mesh areas—i.e., every curve or mesh area will be a different zone in ANSYS FLUENT. You may also create the groups manually, generating groups consisting of all nodes related to a given curve (2D) or mesh area (3D).

6.3.6.3. Grouping Elements to Create Cell Zones

Elements in I-deas are grouped using the `Group` command to create the multiple cell zones. All elements grouped together are placed in a single cell zone in ANSYS FLUENT. If the elements are not grouped, ANSYS FLUENT will place all the cells into a single zone.

6.3.6.4. Deleting Duplicate Nodes

I-deas may generate duplicate or coincident nodes in the process of creating elements. These nodes must be removed in I-deas before writing the universal file for import into ANSYS FLUENT.

6.3.7. NASTRAN Files

There are three different ways in which you can import a NASTRAN file into ANSYS FLUENT:

- You can generate a NASTRAN surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, and/or hexahedral elements, and import it into TGrid using the commands described in the TGrid User's Guide and adhering to the restrictions described in Appendix B of the TGrid User's Guide. In TGrid, complete the mesh generation (if necessary) and then follow the instructions in *TGrid Mesh Files* (p. 149).
- You can generate a NASTRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, and import it directly using the **File/Import/NASTRAN** menu item, as described in *NASTRAN Files* (p. 81).
- You can generate a NASTRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Then use the `fe2ram` filter to convert the NASTRAN file to the format used by ANSYS FLUENT. To convert an input file in NASTRAN format to an output file in ANSYS FLUENT format, follow the instructions below in *Using the fe2ram Filter to Convert Files* (p. 155). After the output file has been written, you can read it into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

After reading a triangular or tetrahedral NASTRAN volume mesh using the latter methods perform smoothing and swapping (as described in *Improving the Mesh by Smoothing and Swapping* (p. 1349)) to improve its quality.

6.3.7.1. Recognized NASTRAN Bulk Data Entries

The following NASTRAN file datasets are recognized by the ANSYS FLUENT mesh import utility:

- **GRID** single-precision node coordinates
- **GRID*** double-precision node coordinates
- **CBAR** line elements
- **CTETRA, CTRIA3** tetrahedral and triangular elements
- **CHEXA, CQUAD4, CPENTA** hexahedral, quadrilateral, and wedge elements

For 2D volume meshes, the elements must exist in a constant z plane.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.7.2. Deleting Duplicate Nodes

NASTRAN may generate duplicate or coincident nodes in the process of creating elements. These nodes must be removed in NASTRAN before writing the file for import into ANSYS FLUENT.

6.3.8. PATRAN Neutral Files

There are three different ways in which you can import a PATRAN Neutral file into ANSYS FLUENT.

- You can generate a PATRAN surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, and/or hexahedral elements, and import it into TGrid using the commands described in the TGrid User's Guide and adhering to the restrictions described in Appendix B of the TGrid User's Guide. In TGrid, complete the mesh generation (if necessary) and then follow the instructions in *TGrid Mesh Files* (p. 149).
- You can generate a PATRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements (grouping nodes with the same component-group name) and import it directly to ANSYS FLUENT by selecting the **File/Import/PATRAN** menu item, as described in *PATRAN Neutral Files* (p. 82).
- You can generate a PATRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements and then use the `fe2ram` filter to convert the Neutral file into the format used by ANSYS FLUENT. To convert an input file in PATRAN Neutral format to an output file in ANSYS FLUENT format, follow the instructions below in *Using the fe2ram Filter to Convert Files* (p. 155). After the output file has been written, you can read it into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

After reading a triangular or tetrahedral PATRAN volume mesh using the latter methods perform smoothing and swapping (as described in *Improving the Mesh by Smoothing and Swapping* (p. 1349)) to improve its quality.

Important

To retain a zone type during PATRAN Neutral file import, please add the abbreviated form of the "zone-type" before the "zone-name", as shown below:

Zone Type in ANSYS FLUENT	Zone Name for PATRAN export (zone-type)-(zone-name)
mass-flow-inlet	m-f-i-zone-name
pressure-outlet	p-o-zone-name
fan	fan-zone-name
velocity-inlet	v-i-zone-name
pressure-far-field	p-f-f-zone-name
solid	solid-zone-name
fluid	fluid-zone-name

6.3.8.1. Recognized PATRAN Datasets

The following PATRAN Neutral file packet types are recognized by the ANSYS FLUENT mesh import utility:

- **Node Data** Packet Type 01
- **Element Data** Packet Type 02

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Distributed Load Data** Packet Type 06
- **Node Temperature Data** Packet Type 10
- **Name Components** Packet Type 21
- **File Header** Packet Type 25

For 2D volume meshes, the elements must exist in a constant z plane.

6.3.8.2. Grouping Elements to Create Cell Zones

Elements are grouped in PATRAN using the `Named Component` command to create the multiple cell zones. All elements grouped together are placed in a single cell zone in ANSYS FLUENT. If the elements are not grouped, ANSYS FLUENT will place all the cells into a single zone.

6.3.9. Mechanical APDL Files

There are three different ways in which you can import an Mechanical APDL file into ANSYS FLUENT.

- You can generate a surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements using Mechanical APDL or ARIES, and import it into TGrid using the commands described in the TGrid User's Guide and adhering to the restrictions described in Appendix B of the TGrid User's Guide. In TGrid, complete the mesh generation (if necessary) and then follow the instructions in *TGrid Mesh Files* (p. 149).
- You can generate a Mechanical APDL volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, as well as with higher order elements like 20 node hexahedron, SOLID92, and SOLID187. Then import it directly to ANSYS FLUENT using the **File/Import/Mechanical APDL** menu item, as described in *Mechanical APDL Files* (p. 81).

  The higher order elements will be converted to their corresponding linear elements during the import in ANSYS FLUENT.
- You can generate a Mechanical APDL volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, and then use the `fe2ram` filter to convert the Mechanical APDL file into the format used by ANSYS FLUENT. To convert an input file in **ANSYS** 5.4 or 5.5 format to an output file in ANSYS FLUENT format, follow the instructions in *Using the fe2ram Filter to Convert Files* (p. 155). After the output file has been written, you can read it into ANSYS FLUENT using the **File/Read/Mesh...** menu item, as described in *Reading Mesh Files* (p. 62).

After reading a triangular or tetrahedral volume mesh using method 2 or 3 above, you should perform smoothing and swapping (as described in *Improving the Mesh by Smoothing and Swapping* (p. 1349)) to improve its quality.

6.3.9.1. Recognized ANSYS 5.4 and 5.5 Datasets

ANSYS FLUENT can import mesh files from **ANSYS** 5.4 and 5.5 (`.cdb` files), retaining original boundary names. The following **ANSYS** file datasets are recognized by the ANSYS FLUENT mesh import utility:

- **NBLOCK** node block data
- **EBLOCK** element block data
- **CMBLOCK** element/node grouping

The elements must be STIF63 linear shell elements. In addition, if element data without an explicit element ID is used, the filter assumes sequential numbering of the elements when creating the zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.10. ANSYS CFX Files

You can import the meshes from 3D ANSYS CFX files, such as definition (`.def`) and result (`.res`) files into ANSYS FLUENT, using the **File/Import/CFX** menu item, as described in *ANSYS CFX Files* (p. 77). The `fe2ram` utility is used as the import filter, which can be used as a stand-alone program to obtain an ANSYS FLUENT mesh file. See *Using the fe2ram Filter to Convert Files* (p. 155) for information about `fe2ram`.

Important

Note that you have the ability to import only the mesh from a ANSYS CFX file, and not any results or data.

When importing a mesh from an ANSYS CFX definition or results file, select whether you want the ANSYS FLUENT zones to be created from the physics data objects or the primitive mesh region objects. The former option is the default and is called "Zoning by CCL" and the latter option is called "Zoning by Group". This will allow you to choose the type of mesh topology you would like to preserve when importing the file:

- If you want zones to be created from the physics data objects, enable **Create Zones from CCL Physics Data** in the **Select File** dialog box.
- If you want zones to be created from the primitive mesh region objects, disable **Create Zones from CCL Physics Data** in the **Select File** dialog box. This will result in the group zoning.

Important

The primitive mesh topology may contain additional regions which do not appear in the physics definition.

The default import method is the **Create Zones from CCL Physics Data** method. This method will not import ANSYS CFX subdomain regions, however, the zoning by group method can import subdomain regions.

The 3D element set corresponding to zones/domains present in these files are imported as cell zones in ANSYS FLUENT. They may contain tetrahedral, pyramidal, wedge, and hexahedral elements. The boundary zones in these files are a group of faces with a boundary condition name/type and are imported as face zones with the boundary condition name/type retained in ANSYS FLUENT. The following boundary condition types are retained:

- **inlet**
- **outlet**
- **symmetry**
- **interface**
- **wall**

The boundaries of type `Interface` may be conformal or non-conformal. If they are non-conformal, they are retained. However, conformal interfaces contain coincident nodes which are merged and changed to type `Interior`. For some cases, for the merge to work correctly, the merge tolerance may need to be adjusted. Alternatively, the *Fuse Face Zones Dialog Box* (p. 2080) in ANSYS FLUENT can be used to merge the conformal interfaces. For details, see *Fusing Face Zones* (p. 190).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.11. Using the fe2ram Filter to Convert Files

The `fe2ram` filter can be used to manually convert files of certain formats into ANSYS FLUENT mesh files, which can then be read into ANSYS FLUENT. To use the `fe2ram` filter, enter the following at a command prompt in a terminal or command window:

`utility fe2ram` [*dimension*] *format* [*zoning*] *input_file output_file*

Note

The items enclosed in square brackets are optional. Do not type the square brackets.

- *dimension* indicates the dimension of the dataset. Replace *dimension* with `-d2` to indicate that the mesh is two dimensional. For a 3D mesh, do not enter any value for *dimension*, because 3D is the default.
- *format* indicates the format of the file you wish to convert. For example, replace *format* with `-tANSYS` for a Mechanical APDL file, `-tIDEAS` for an I-deas file, `-tNASTRAN` for a NASTRAN file, etc. To print a list of the formats which `fe2ram` can convert, type

  `utility fe2ram -cl -help`.
- *zoning* indicates how zones were identified in the original format. Replace *zoning* by `-zID` for a mesh that was zoned by property IDs, or `-zNONE` to ignore all zone groupings. For a mesh zoned by group, do not enter anything for *zoning*, because zoning by groups is the default.
- *input_file* is the name of the original file. *output_file* is the name of the file to which you want to write the converted mesh information.

  For example, if you wanted to convert the 2D I-deas volume mesh file `sample.unv` to an output file called `sample.grd`, you will enter the following command:

  ```
  utility fe2ram -d2 -tIDEAS sample.unv sample.grd
  ```

6.3.12. Using the tpoly Filter to Remove Hanging Nodes

While ANSYS FLUENT can accept meshes that contain hanging nodes, the creation of interior walls can yield an error if hanging nodes are located on the zone that is turned into an interior wall. This problem can occur in the following cases:

- If you read a mesh file that has an interior (or two-sided) wall boundary condition setup, but the shadow wall hasn't been created yet.
- If you turn an interior surface into a wall.
- If you slit an interior surface (e.g., turn it into a non-conformal interface).

Such error-producing hanging nodes may be present in hexcore meshes, for example. You can remove the hanging nodes by converting the transitional hexahedral cells of the hexcore mesh to polyhedra. Each transitional cell that is converted will retain the same overall dimensions, but will have more than the original 6 faces. The conversion to polyhedra should take place prior to reading the mesh into ANSYS FLUENT, and can be done using either the tpoly filter or TGrid.

When you use the `tpoly` filter, you must specify an input case file that contains a hexcore mesh. This file can either be in ASCII or Binary format, and the file should be unzipped. If the input file does not contain a hexcore mesh, then none of the cells are converted to polyhedra. When you use the `tpoly` filter, you should

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specify an output case file name. After the input file has been processed by the `tpoly` filter, an ASCII output file is generated.

Important

The output case file resulting from a `tpoly` conversion only contains mesh information. None of the solver-related data of the input file is retained.

To convert a file using the `tpoly` filter, before starting ANSYS FLUENT, type the following:

`utility tpoly` *input_filename output_filename*

You can also use TGrid to convert the transitional cells to polyhedra. You must either read in or create the hexcore mesh in TGrid, and then save the mesh as a case file with polyhedra. To do this, use the **File/Write/Case...** menu option, being sure to enable the **Write As Polyhedra** option in the **Select File** dialog box.

6.3.12.1. Limitations

Converted hexcore meshes have the following limitations:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - `extrude-face-zone` under the `modify-zone` option
  - fuse
  - skewness smoothing
  - swapping (will not affect polyhedral cells)
- The polyhedral cells that result from the conversion are not eligible for adaption. For more information about adaption, see *Adapting the Mesh* (p. 1325).

6.3.13. FLUENT/UNS and RAMPANT Case Files

If you have a FLUENT/UNS 3 or 4 case file or a RAMPANT 2, 3, or 4 case file and you want to run an ANSYS FLUENT simulation using the same mesh, you can read it into ANSYS FLUENT using the **File/Read/Case...** menu item, as described in *Reading FLUENT/UNS and RAMPANT Case and Data Files* (p. 70).

6.3.14. FLUENT 4 Case Files

If you have a FLUENT 4 case file and you want to run an ANSYS FLUENT simulation using the same mesh, import it into ANSYS FLUENT using the **File/Import/FLUENT 4 Case File...** menu item, as described in *FLUENT 4 Case Files* (p. 83). ANSYS FLUENT will read mesh information and zone types from the FLUENT 4 case file.

Important

FLUENT 4 may interpret some pressure boundaries differently from the current release of ANSYS FLUENT. Check the conversion information printed out by ANSYS FLUENT to see if you need to modify any boundary types.

To manually convert an input file in FLUENT 4 format to an output file in the current ANSYS FLUENT format, enter the following command:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`utility fl42seg` *input_filename output_filename*

After the output file has been written, you can read it into ANSYS FLUENT using the **File/Read/Case...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.15. ANSYS FIDAP Neutral Files

If you have an ANSYS FIDAP Neutral file and you want to run an ANSYS FLUENT simulation using the same mesh, import it using the menu item, as described in *ANSYS FIDAP Neutral Files* (p. 79). ANSYS FLUENT will read mesh information and zone types from the ANSYS FIDAP file.

To manually convert an input file in ANSYS FIDAP format to an output file in ANSYS FLUENT

`utility fe2ram [`*dimension*`] -tFIDAP7` *input_file output_file*

The item in square brackets is optional. Do not type the square brackets. For a 2D file, replace *dimension* with `-d2`. For a 3D file, do not enter anything for *dimension*, because 3D is the default.

After the output file has been written, read it into ANSYS FLUENT using the **File/Read/Case...** menu item, as described in *Reading Mesh Files* (p. 62).

6.3.16. Reading Multiple Mesh/Case/Data Files

There may be some cases in which you will need to read multiple mesh files (subdomains) to form your computational domain.

- To solve on a multiblock mesh, generate each block of the mesh in the mesh generator and save it to a separate mesh file.
- For very complicated geometries, it may be more efficient to save the mesh for each part as a separate mesh file.

The mesh node locations need not be identical at the boundaries where two separate meshes meet. ANSYS FLUENT can handle non-conformal mesh interfaces. See *Non-Conformal Meshes* (p. 161) for details about non-conformal mesh boundaries.

There are two ways for reading multiple mesh files in ANSYS FLUENT:

- Using ANSYS FLUENT's ability to read multiple mesh files.
- Using TGrid or `tmerge`.

6.3.16.1. Using ANSYS FLUENT's Ability to Read Multiple Mesh Files

ANSYS FLUENT allows you to handle more than one mesh at a time within the same solver settings. This capability of handling multiple meshes saves time, since you can directly read in the different mesh files in ANSYS FLUENT itself without using other tools like TGrid or `tmerge`.

The steps to take when reading more than one mesh file are:

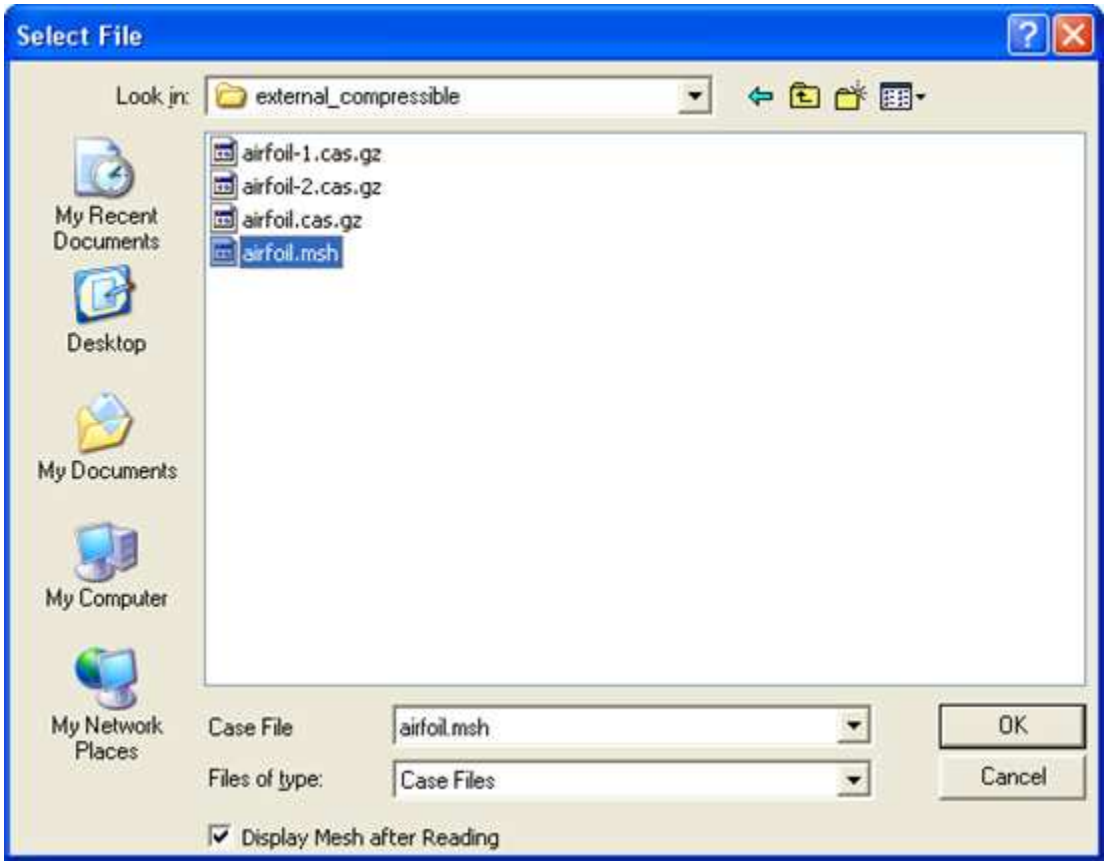
1. Read in your first mesh file.

   File → **Read** → **Mesh...**

   In *The **Select File** Dialog Box* (p. 33) (*Figure 6.24* (p. 158)), select the mesh file and click **OK**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.24 The Select File Dialog Box



2. Read in your second mesh file and append it to the first mesh selected in the first step.

   Mesh → **Zone** → **Append Case File...**

   In *The **Select File** Dialog Box* (p. 33), select the second mesh file and click **OK**.

3. (optional). Display your meshes using the **Mesh Display** dialog box.

   General → **Display...**

   You will find that the second mesh is appended to the first.

ANSYS FLUENT also allows you to append the data on the mesh. To do that, follow the procedure above. For the second step, use the following menu item:

Mesh → **Zone** → **Append Case & Data Files...**

Select the case file in *The **Select File** Dialog Box* (p. 33) (*Figure 6.24* (p. 158)), and click **OK**. Both the case and data files will be appended.

Important

Reading multiple mesh and data options are available for serial and parallel cases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.16.2. Using TGrid or tmerge

1. Generate the mesh for the whole domain in the mesh generator, and save each cell zone (or block or part) to a separate mesh file for ANSYS FLUENT.

   Important

   If one (or more) of the meshes you wish to import is structured (e.g., a FLUENT 4 mesh file), first convert it to ANSYS FLUENT format using the `fl42seg` filter described in *FLUENT 4 Case Files* (p. 156).

2. Before starting the solver, use either TGrid or the `tmerge` filter to combine the meshes into one mesh file. The TGrid method is convenient, but the `tmerge` method allows you to rotate, scale, and/or translate the meshes before they are merged. Note that the `tmerge` filter allows you to merge large meshes with very low memory requirement.

   - To use TGrid, do the following:

     a. Read all of the mesh files into TGrid. When TGrid reads the mesh files, it will automatically merge them into a single mesh.

     b. Save the merged mesh file.

     See the TGrid User's Guide for information about reading and writing files in TGrid.

   - To use the `tmerge` filter, do the following before starting ANSYS FLUENT:

     a. For 3D problems, type `utility tmerge -3d`. For 2D problems, type `utility tmerge -2d`.

     b. When prompted, specify the names of the input files (the separate mesh files) and the name of the output file in which to save the complete mesh. Be sure to include the `.msh` extension.

     c. For each input file, specify scaling factors, translation distances, and rotation information.

     For information about the various options available when using `tmerge`, type `utility tmerge -h`.

3. Read the combined mesh file into the solver in the usual manner (using the **File/Read/Mesh...** menu item).

For a conformal mesh, if you do not want a boundary between the adjacent cell zones, use the **Fuse Face Zones** dialog box to fuse the overlapping boundaries. For details, see *Fusing Face Zones* (p. 190). The matching faces will be moved to a new zone with a boundary type of **interior**. If all faces on either or both of the original zones have been moved to the new zone, the original zone(s) will be discarded.

Important

If you are planning to use sliding meshes, or if you have non-conformal boundaries between adjacent cell zones, do not combine the overlapping zones. Instead, change the type of the two overlapping zones to **interface** (as described in *Non-Conformal Meshes* (p. 161)).

In this example, scaling, translation, or rotation is not requested. Hence you can simplify the inputs to the following:

```
user@mymachine:> utility tmerge -2d
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Starting /Fluent.Inc/utility/tmerge2.1/ultra/tmerge_2d.2.1.13

 Append 2D grid files.
 tmerge2D Fluent Inc, Version 2.1.11

 Enter name of grid file (ENTER to continue) :  my1.msh


 x,y scaling factor, eg. 1 1          : 1 1


 x,y translation, eg. 0 1             : 0 0


 rotation angle (deg), eg. 45         : 0


 Enter name of grid file (ENTER to continue) :  my2.msh


 x,y scaling factor, eg. 1 1          : 1 1


 x,y translation, eg. 0 1             : 0 0


rotation angle (deg), eg. 45          : 0


 Enter name of grid file (ENTER to continue) :  Enter


 Enter name of output file            : final.msh


 Reading...
  node zone: id 1, ib 1, ie 1677, typ 1
  node zone: id 2, ib 1678, ie 2169, typ 2
    .
    .
    .
 done.
 Writing...
  492 nodes, id 1, ib 1678, ie 2169, type 2.
 1677 nodes, id 2, ib 1, ie 1677, type 1.
    .
    .
    .
 done.
 Appending done.
```

6.3.17. Reading Surface Mesh Files

Surface meshes are used as background meshes for geometry-based adaption. Perform the following steps to read the surface mesh file into ANSYS FLUENT:

1. Open the **Geometry Based Adaption** dialog box.

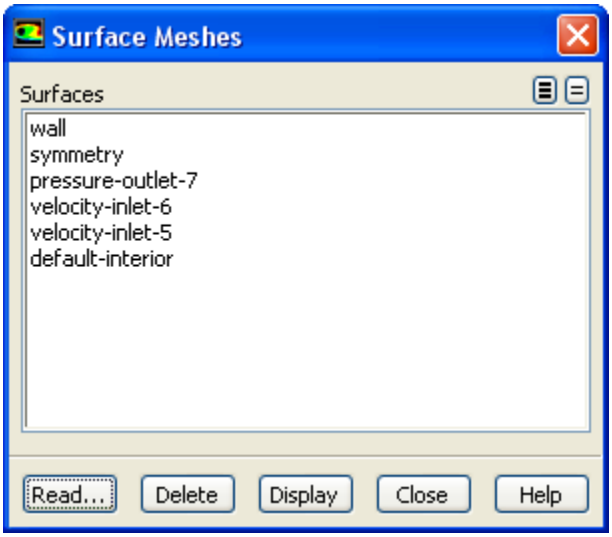
   Adapt → **Geometry...**

2. Enable the **Reconstruct Geometry** option.
3. Click the **Surface Meshes...** button to open the **Surface Meshes** dialog box (*Figure 6.25* (p. 161)).
4. In the *Surface Meshes Dialog Box* (p. 2132), click **Read...** and select the surface mesh file using *The **Select File** Dialog Box* (p. 33).

   Note that you can also display and delete the surfaces using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.25 The Surface Meshes Dialog Box



6.4. Non-Conformal Meshes

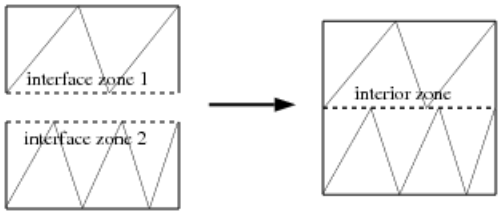
In ANSYS FLUENT it is possible to use a mesh that has non-conformal interfaces, that is, boundaries between cell zones in which the mesh node locations are not identical. Such non-conformal interfaces permit the cell zones to be easily connected to each other by passing fluxes from one mesh to another. The principle requirement is that the boundary zones that comprise the non-conformal interface must overlap either partially or fully. (This requirement does not apply to non-conformal periodic boundaries.)

6.4.1. Non-Conformal Mesh Calculations
6.4.2. Non-Conformal Interface Algorithm
6.4.3. Requirements and Limitations of Non-Conformal Meshes
6.4.4. Using a Non-Conformal Mesh in ANSYS FLUENT

6.4.1. Non-Conformal Mesh Calculations

To compute the flux across the non-conformal boundary, ANSYS FLUENT must first compute the intersection between the interface zones that comprise the boundary. The resulting intersection produces an interior zone where the two interface zones overlap (see *Figure 6.26* (p. 161)).

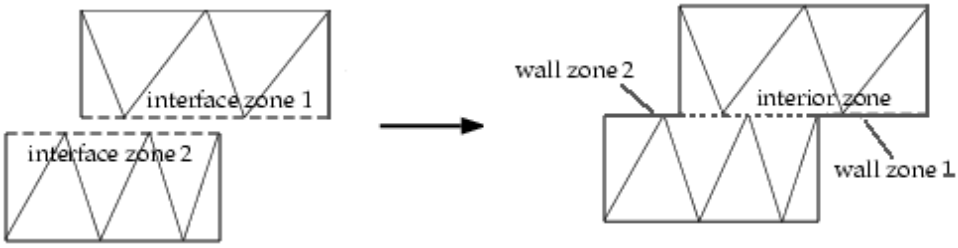
Figure 6.26 Completely Overlapping Mesh Interface Intersection



If one of the interface zones extends beyond the other (*Figure 6.27* (p. 162)), by default ANSYS FLUENT will create additional wall zones for the portion(s) of the boundary where the two interface zones do not overlap.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

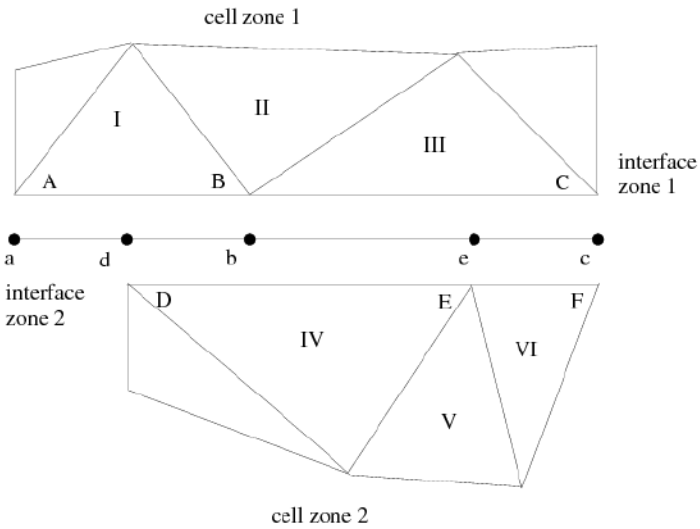
Figure 6.27 Partially Overlapping Mesh Interface Intersection



Fluxes across the mesh interface are computed using the faces resulting from the intersection of the two interface zones, not from the interface zone faces.

In the example shown in *Figure 6.28* (p. 162), the interface zones are composed of faces A-B and B-C, and faces D-E and E-F.

Figure 6.28 Two-Dimensional Non-Conformal Mesh Interface



The intersection of these zones produces the faces a-d, d-b, b-e, and e-c. Faces produced in the region where the two cell zones overlap (d-b, b-e, and e-c) are grouped to form an interior zone, while the remaining face (a-d) forms a wall zone.

To compute the flux across the interface into cell IV, face D-E is ignored and instead faces d-b and b-e are used to bring information into cell IV from cells I and III.

While the previous discussion described the default treatment of a non-conformal interface, there are several options you can enable to revise the treatment of the fluxes at the interface:

- periodic boundary condition
- periodic repeats
- coupled wall

These non-conformal interface options are described in the following sections.

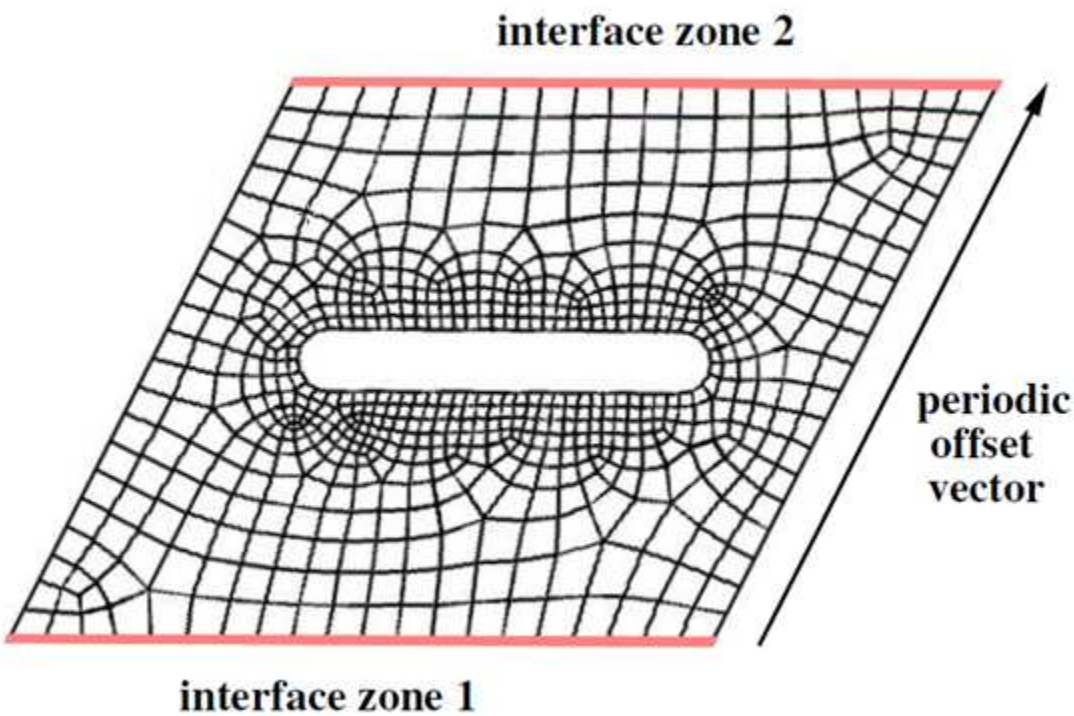
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1.1. The Periodic Boundary Condition Option

Non-conformal interfaces can be used to implement a periodic boundary condition like that described for conformal periodic boundaries (see *Periodic Boundary Conditions* (p. 321)). The advantage of using a mesh interface is that, unlike the standard periodic boundary condition, the nodes of the two zones do not have to match one-for-one.

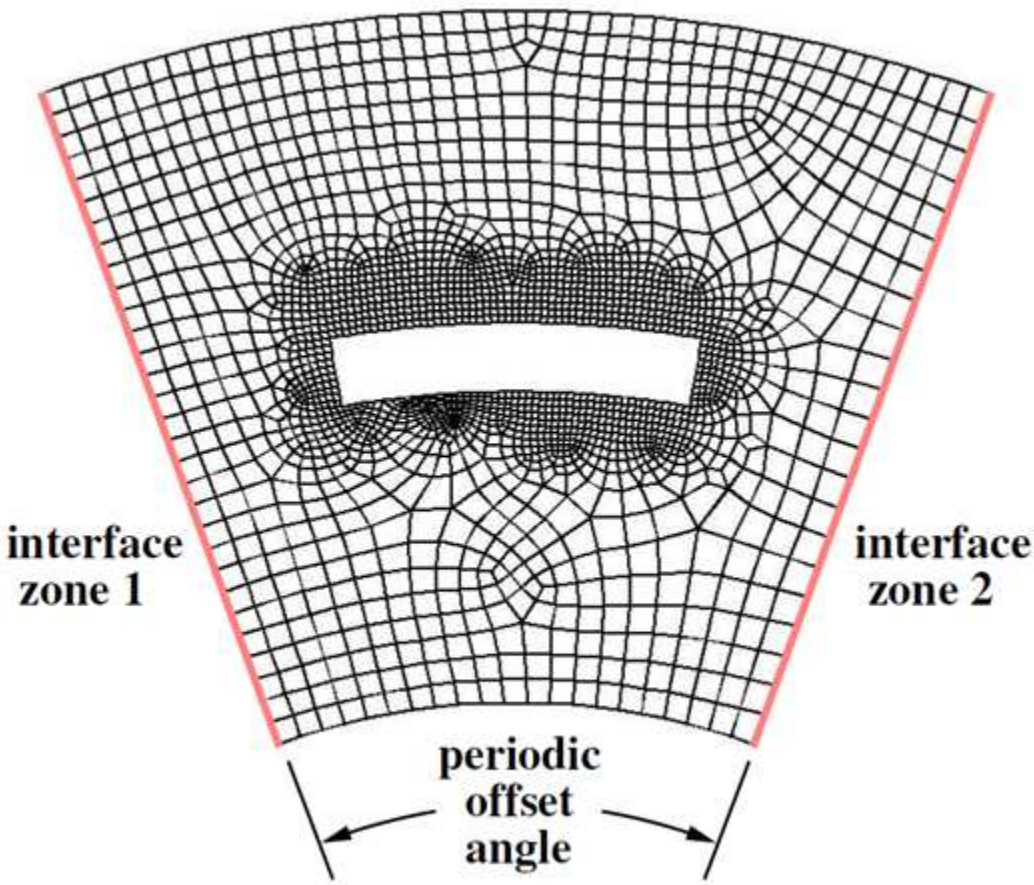
The interface zones that utilize the periodic boundary condition option (*Figure 6.29* (p. 163) and *Figure 6.30* (p. 164)) are coupled in the manner described in the previous section, except that the zones do not overlap (i.e., the zones are not spatially coincident at any point). In order to generate the new faces that will be used to compute the fluxes across the interface, the nodes of the first zone are either translated or rotated (about a given axis) onto the other zone. The distance / angle that the nodes are translated / rotated is called the "periodic offset". The new faces will be defined between all of the combined nodes, and then applied to each of the original zones.

Figure 6.29 Non-Conformal Periodic Boundary Condition (Translational)



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.30 Non-Conformal Periodic Boundary Condition (Rotational)

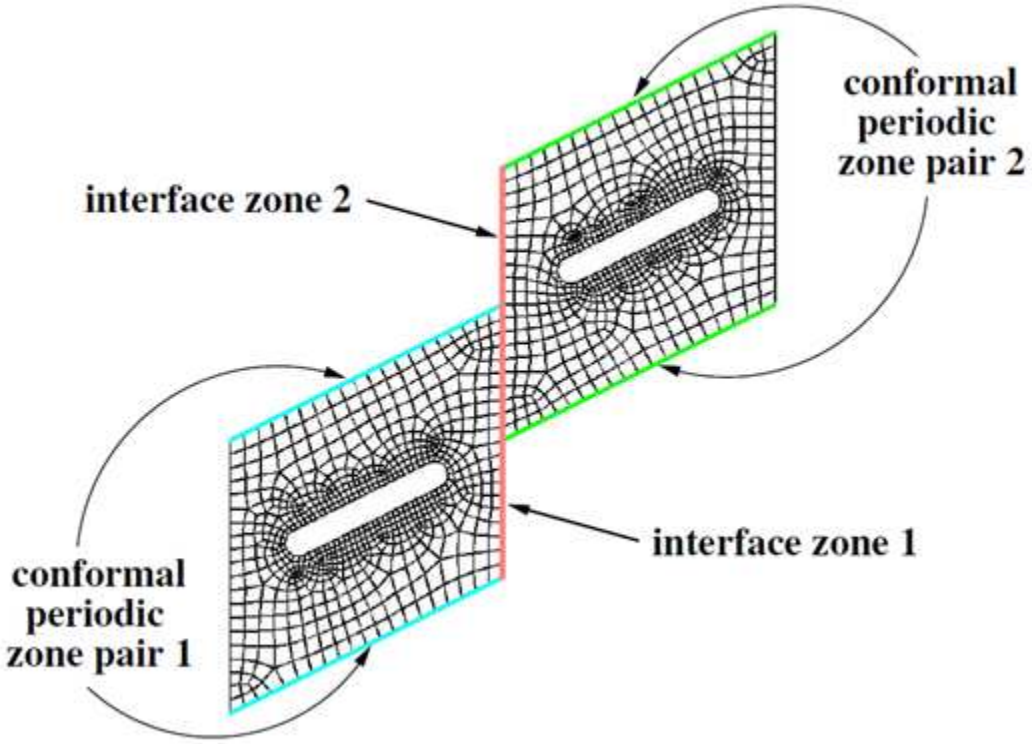


6.4.1.2. The Periodic Repeats Option

The periodic repeats option is appropriate when each of the interface zones is adjacent to a pair of conformal periodic zones (see *Figure 6.31* (p. 165) and *Figure 6.32* (p. 166)). The periodic repeats option takes into account the repeating nature of the flow solutions in the two cell zones in the following manner. Wherever the interface zones overlap (i.e., wherever interface zone 1 and 2 are spatially coincident), the fluxes on either side of the interface are coupled in the usual way. The portion of interface zone 1 that does not overlap is coupled to the non-overlapping portion of interface zone 2, by translating or rotating the fluxes by the periodic offset. This is similar to the treatment of non-conformal periodic boundary conditions. The periodic repeats option is typically used in conjunction with the sliding mesh model when simulating the interface between a rotor and stator.

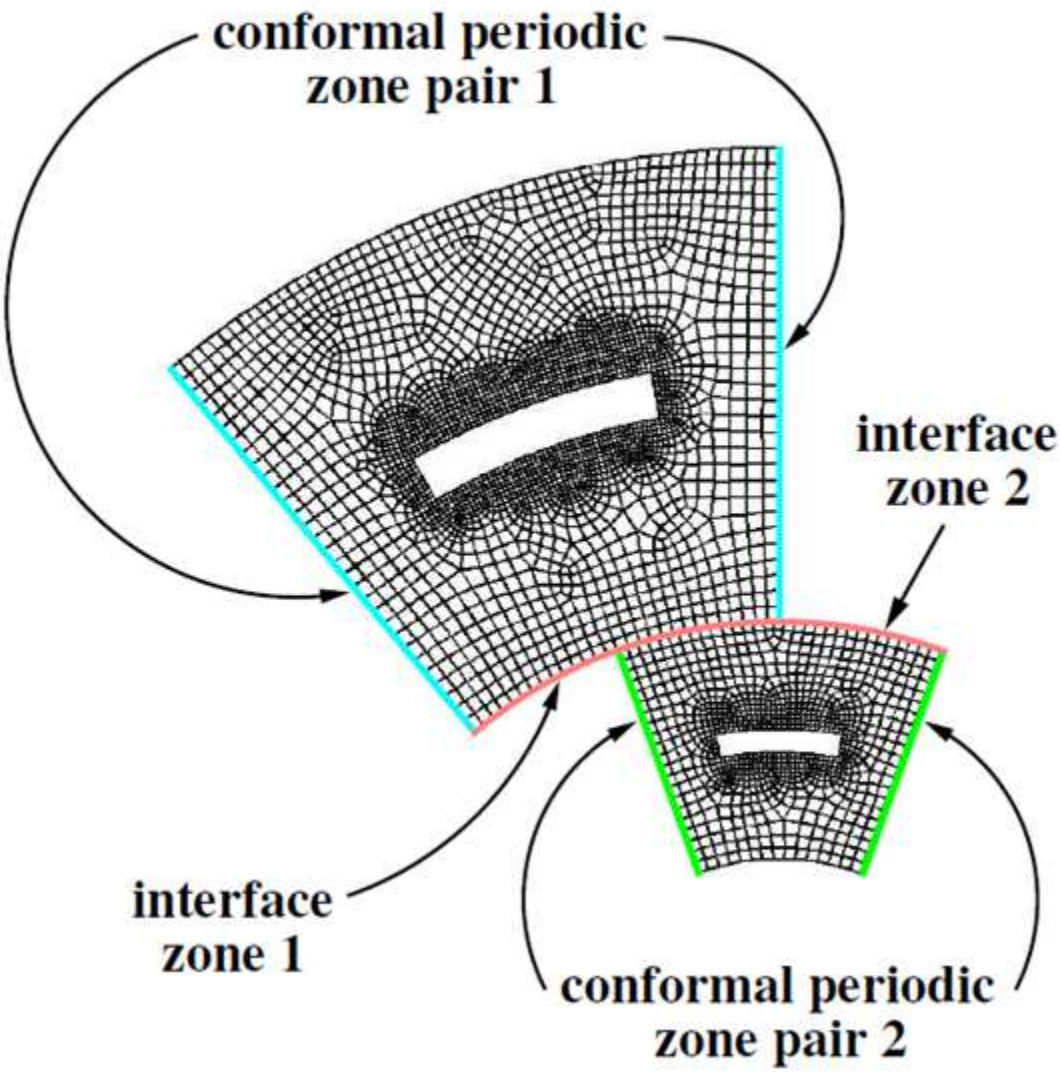
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.31 Translational Non-Conformal Interface with the Periodic Repeats Option



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.32 Rotational Non-Conformal Interface with the Periodic Repeats Option



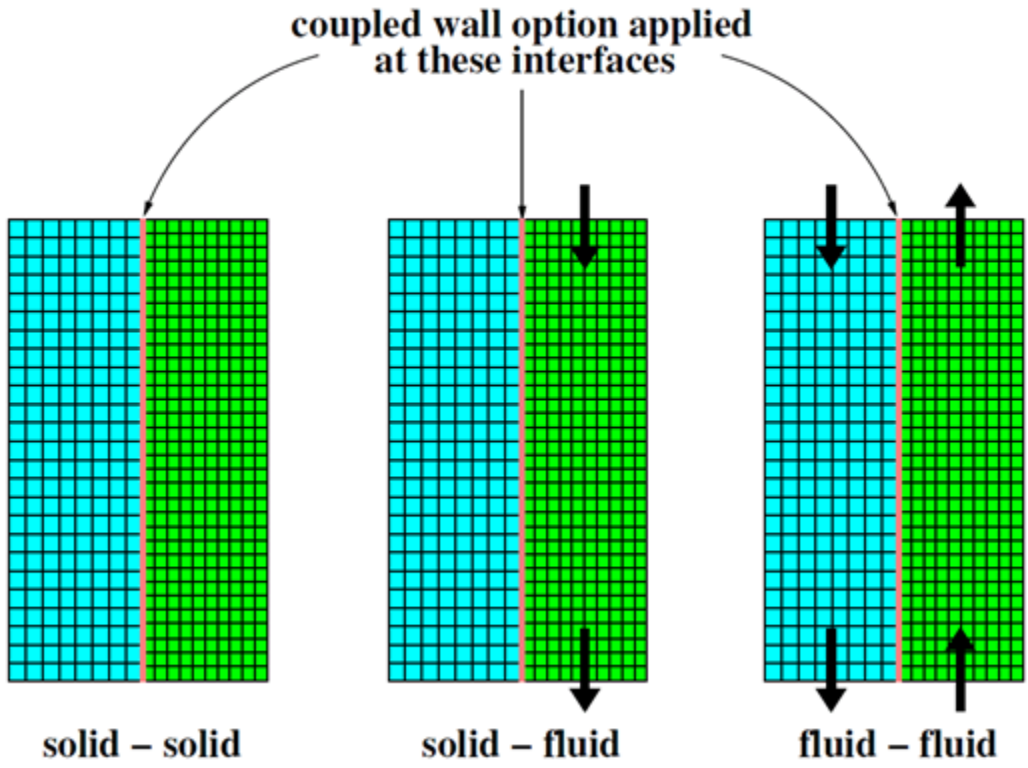
6.4.1.3. The Coupled Wall Option

As described previously, the typical function of non-conformal interfaces is to couple fluid zones, so as to permit fluid flow to pass from one mesh interface to the other. Another available option is to create a coupled wall boundary at the interface. In such a case, fluid flow would not pass across the interface, as the interface is acting as a wall zone. Coupled wall heat transfer, on the other hand, would be permitted. Such an interface is required if one or both of the cell zones is a solid. It is also allowable if both of the cell zones are fluids; for example, you can model a thin wall or baffle separating the two fluid zones. *Figure 6.33* (p. 167) illustrates coupled walls with both solid and fluid zones.

Note that coupled walls can also make use of the periodic repeats option. That is, both options can be invoked simultaneously. For details see *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.33 Non-Conformal Coupled Wall Interfaces



6.4.2. Non-Conformal Interface Algorithm

In the current version of ANSYS FLUENT, non-conformal interface calculations are handled using a virtual polygon approach, which stores the area vector and centroid of the polygon faces. This approach does not involve node movement and cells are not necessarily water-tight cells. Hence gradients are corrected to take into account the missing face area. Note that in transient cases, the stationary non-conformal interfaces will be automatically preserved by ANSYS FLUENT during mesh update.

Previous versions of FLUENT (6.1 or earlier) used a triangular face approach, which triangulated the polygon intersection faces and stored triangular faces. This approach involved node movement and water-tight cells, and was not as stable as the current virtual polygon approach. Note that case files in which the interface was set up using ANSYS FLUENT 6.1 or earlier can be read and run normally in the current version of ANSYS FLUENT, which will use the virtual polygon approach rather than the triangular face approach.

It is possible that distorted meshes may be produced during sliding mesh calculations, generating what are called "left-handed" faces. You cannot obtain a flow solution until all of the faces are "right handed", and so ANSYS FLUENT corrects the left handedness of these faces automatically. In extreme cases, the left-handed faces cannot be fully corrected and are deleted automatically, so that the solution does not diverge.

Left-handed cells can also be created for the geometries that contain sharp edges and corners, which may affect the final solution. For such geometries, it is recommended to first separate the zones and then create the interfaces separately to get the better solution.

The additional input of the angle / translation vector at the `angle/translation-vector` prompt in the console may be required to recreate face-periodic interfaces. Also, with the current mesh interface algorithm in parallel, there is no need for encapsulation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.3. Requirements and Limitations of Non-Conformal Meshes

This section describes the requirements and limitations of non-conformal meshes:

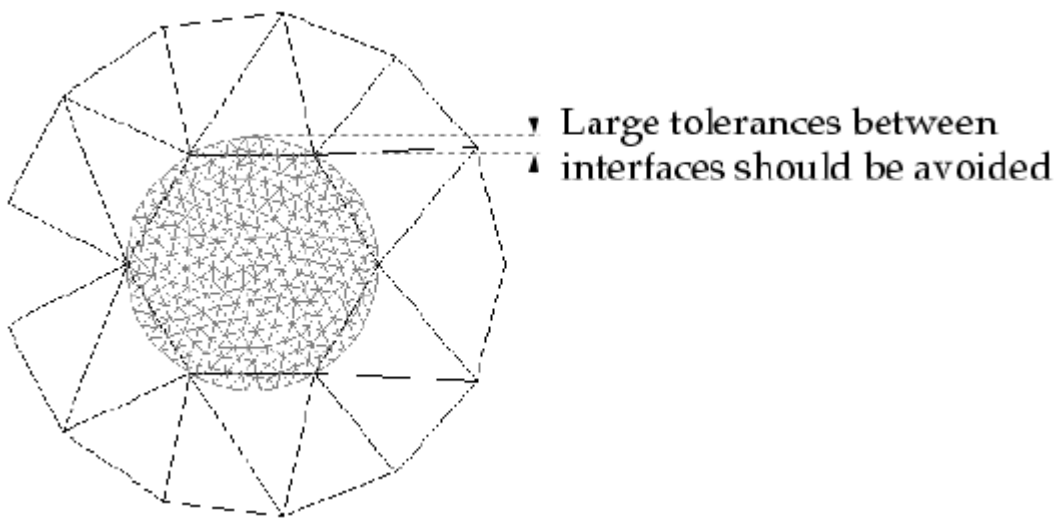
- The mesh interface can be of any shape (including a non-planar surface, in 3D), provided that the two interface boundaries are based on the same geometry. If there are sharp features (e.g., 90-degree angles) or curvature in the mesh, it is especially important that both sides of the interface closely follow that feature.

  For example, consider the case of two concentric circles that define two fluid zones with a circular, non-conformal interface between them, as shown in *Figure 6.34* (p. 168). Because the node spacing on the interface edge of the outer fluid zone is coarse compared to the radius of curvature, the interface does not closely follow the feature (in this case, the circular edge.)

  Important

  The maximum tolerance between two interfaces should not be larger than their adjacent cell size at that location That is no cell should be completely enclosed between two interfaces.

  Figure 6.34 A Circular Non-Conformal Interface



- If you create a single mesh with multiple cell zones separated by a non-conformal boundary, you must be sure that each cell zone has a distinct face zone on the non-conformal boundary.

  The face zones for two adjacent cell zones will have the same position and shape, but one will correspond to one cell zone and one to the other. It is also possible to create a separate mesh file for each of the cell zones, and then merge them as described in *Reading Multiple Mesh/Case/Data Files* (p. 157).
- All periodic zones must be correctly oriented (either rotational or translational) before you create the non-conformal interface.
- You must not enable the periodic boundary condition option if the interface is adjacent to another non-conformal interface.
- In order for the periodic boundary condition option or periodic repeats option to be valid, the edges of the second interface zone must be offset from the corresponding edges of the first interface zone by a uniform amount (either a uniform translational displacement or a uniform rotation angle). This is not true for non-conformal interfaces in general.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The periodic boundary condition option has the additional requirement that the angle associated with a rotational periodic must be able to divide 360 without remainder.

- The periodic repeats option requires that some portion of the two interface zones must overlap (i.e., be spatially coincident).
- The periodic repeats option requires that the non-overlapping portions of the interface zones must have identical shape and dimensions. If the interface is part of a sliding mesh, you must define the mesh motion such that this criterion is met at all times.
- The periodic repeats option requires one pair of conformal periodic zones adjacent to each of the interface zones. For example, when you calculate just one channel and blade of a fan, turbine, etc., you must have conformal periodics on either side of the interface threads. This will not work with non-conformal periodics.

  Note that for 3D cases, you cannot have more than one pair of conformal periodic zones adjacent to each of the interface zones.
- You must not have a single non-conformal interface where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (i.e., the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169) for information about creating coupled interfaces.

6.4.4. Using a Non-Conformal Mesh in ANSYS FLUENT

If your multiple-zone mesh includes non-conformal boundaries, check if the mesh meets all the requirements (listed in *Requirements and Limitations of Non-Conformal Meshes* (p. 168)). This ensures that ANSYS FLUENT can obtain a solution on the mesh. Then do the following:

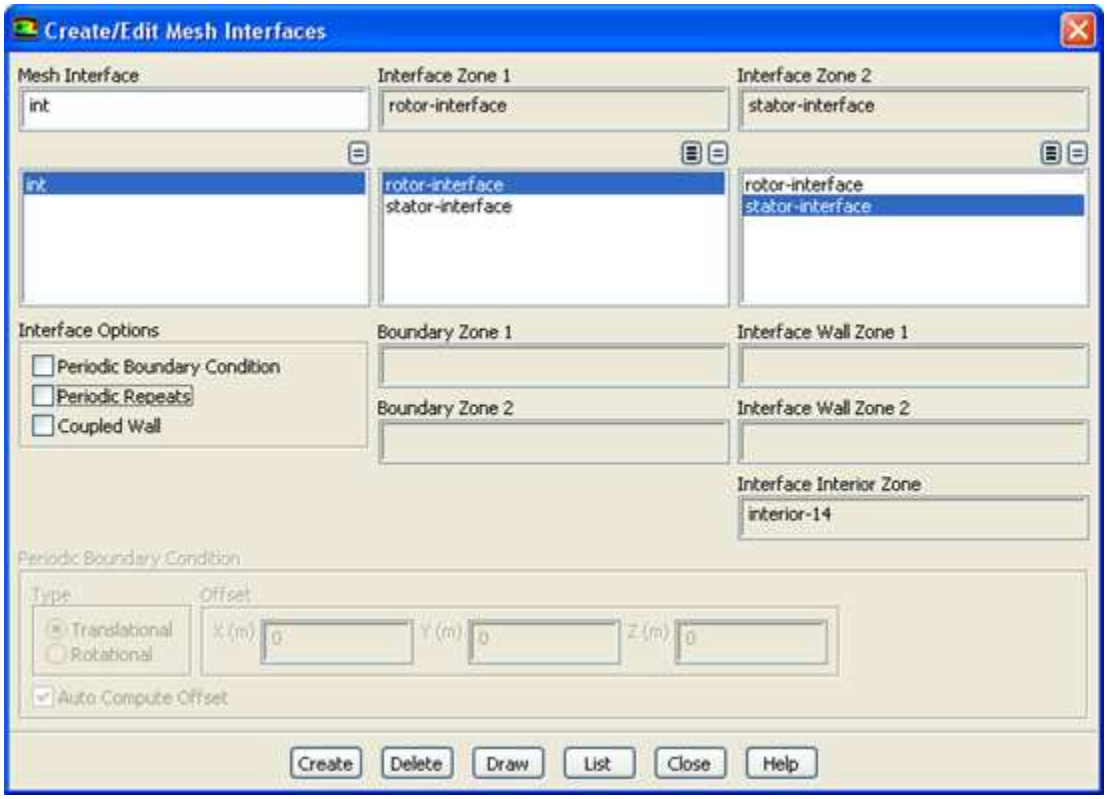
1. Read the mesh into ANSYS FLUENT. If you have multiple mesh files that have not yet been merged, first follow the instructions in *Reading Multiple Mesh/Case/Data Files* (p. 157) to merge them into a single mesh.
2. After reading in the mesh, change the type of each pair of zones that comprises the non-conformal boundary to **interface** (as described in *Changing Cell and Boundary Zone Types* (p. 207)).

   Boundary Conditions
3. Define the non-conformal mesh interfaces in the *Create/Edit Mesh Interfaces Dialog Box* (p. 1876) (*Figure 6.35* (p. 170)).

   Mesh Interfaces → **Create/Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.35 The Create/Edit Mesh Interfaces Dialog Box



a. Enter a name for the interface in the **Mesh Interface** text-entry box.

b. Specify the two interface zones that comprise the mesh interface by selecting one or more zones in the **Interface Zone 1** list and one or more zones in the **Interface Zone 2** list.

Important

If one of your interface zones is much smaller than the other, you should specify the smaller zone as **Interface Zone 1** to improve the accuracy of the intersection calculation.

c. Enable the desired **Interface Options**, if appropriate. There are three options:

- Enable **Periodic Boundary Condition** to create a non-conformal periodic boundary condition interface.
  - Select either **Translational** or **Rotational** as the periodic boundary condition **Type** to define the type of periodicity.
  - Retain the enabled default setting of **Auto Compute Offset** if you want ANSYS FLUENT to automatically compute the offset. After creating the interface, the offsets will be displayed in these fields. The fields will be uneditable when the **Auto Compute Offset** is enabled.
  - Disable **Auto Compute Offset** if you decide that you do not want ANSYS FLUENT to find the offset. In this case, you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Auto computation means that the rotational angle or the translational offset will be automatically calculated and used while creating a non-conformal periodic boundary condition interface. However, it still relies on the **Rotational Axis Origin** and the **Rotational Axis Direction** that was entered for the cell zone in the cell zone condition dialog box (e.g. the **Fluid** dialog box). Therefore, before proceeding with the creation of the non-conformal periodic boundary condition interface, you have to correctly enter the rotational axis for the corresponding cell zone.

Note that auto computation of the non-conformal periodic boundary condition offset does not mean that the **Rotational Axis Origin** and **Rotational Axis Direction** are also detected automatically and updated. It is still your responsibility to set up these values correctly.

Important

The **Periodic Boundary Condition** option is only valid when a single zone is selected in each of the **Interface Zone 1** and the **Interface Zone 2** selection lists.

- Enable **Periodic Repeats** when each of the two cell zones has a single pair of conformal periodics adjacent to the interface (see *Figure 6.31* (p. 165)). This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator.

  Important

  The **Periodic Repeats** option is only valid when a single zone is selected in each of the **Interface Zone 1** and the **Interface Zone 2** selection lists.

- Enable **Coupled Wall** if you would like to model a thermally coupled wall between two fluid zones that share a non-conformal interface.

  Important

  Note that the following interfaces are coupled by default:

  – the interface between a solid zone and fluid zone

  – the interface between a solid zone and solid zone

  Therefore, no action is required in the **Create/Edit Mesh Interfaces** dialog box to set up such interfaces.

d. Click **Create** to create a new mesh interface. For all types of interfaces, ANSYS FLUENT will create boundary zones for the interface (e.g., **wall-9**, **wall-10**), which will appear under **Boundary Zone 1** and **Boundary Zone 2**. If you have enabled the **Coupled** option, ANSYS FLUENT will also create

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall interface zones (e.g., **wall-4**, **wall-4-shadow**), which will appear under **Interface Wall Zone 1** and **Interface Wall Zone 2**.

e. If the two interface zones did not overlap entirely, check the boundary zone type of the zone(s) created for the non-overlapping portion(s) of the boundary. If the zone type is not correct, you can use the *Boundary Conditions Task Page* (p. 1812) to change it.

f. If you have any **Coupled Wall** type interfaces, define boundary conditions (if relevant) by updating the interface wall zones using the **Boundary Conditions** task page.

Boundary Conditions

If you create an incorrect mesh interface, you can select it in the **Mesh Interface** selection list and click the **Delete** button to delete it. Any boundary zones or wall interface zones that were created when the interface was created will also be deleted. You may then proceed with the problem setup as usual.

You can click the **Draw** button to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display interface zones from **Interface Zone 1** or Interface Zone 2 prior to defining any **Mesh Interfaces**. After a **Mesh Interface** is defined, you can select the appropriate mesh interface and click the **Draw** button to display the zones under **Interface Zone 1** and Interface Zone 2 together as defined by the **Mesh Interface**. This is particularly useful if you want to check the location of the interface zones prior to setting up a mesh interface.

6.5. Checking the Mesh

The mesh checking capability in ANSYS FLUENT examines various aspects of the mesh, including the mesh topology, periodic boundaries, simplex counters, and (for axisymmetric cases) node position with respect to the x axis, and provides a mesh check report with details about domain extents, statistics related to cell volume and face area, and information about any problems associated with the mesh. You can check the mesh by clicking the **Check** button in the **General** task page.

General → **Check**

Important

It is generally a good idea to check your mesh right after reading it into the solver, in order to detect any mesh trouble before you get started with the problem setup.

The mesh check examines the topological information, beginning with the number of faces and nodes per cell. A triangular cell (2D) should have 3 faces and 3 nodes, a tetrahedral cell (3D) should have 4 faces and 4 nodes, a quadrilateral cell (2D) should have 4 faces and 4 nodes, and a hexahedral cell (3D) should have 6 faces and 8 nodes. Polyhedral cells (3D) will have an arbitrary number of faces and nodes.

Next, the face handedness and face node order for each zone is checked. The zones should contain all right-handed faces, and all faces should have the correct node order.

The last topological verification is checking the element-type consistency. If a mesh does not contain mixed elements (quadrilaterals and triangles or hexahedra and tetrahedra), ANSYS FLUENT will determine that it does not need to keep track of the element types. By doing so, it can eliminate some unnecessary work.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For axisymmetric cases, the number of nodes below the x axis is listed. Nodes below the x axis are forbidden for axisymmetric cases, since the axisymmetric cell volumes are created by rotating the 2D cell volume about the x axis; thus nodes below the x axis would create negative volumes.

For solution domains with rotationally periodic boundaries, the minimum, maximum, average, and prescribed periodic angles are computed. A common mistake is to specify the angle incorrectly. For domains with translationally periodic boundaries, the boundary information is checked to ensure that the boundaries are truly periodic.

Finally, the simplex counters are verified. The actual numbers of nodes, faces, and cells the solver has constructed are compared to the values specified in the corresponding header declarations in the mesh file. Any discrepancies are reported.

6.5.1. Mesh Check Report
6.5.2. Repairing Meshes

6.5.1. Mesh Check Report

When you click the **Check** button in the **General** task page, a mesh check report will be displayed in the console. The following is a sample of a successful output :

```
  Mesh Check

  Domain Extents:
    x-coordinate: min (m) = -4.000000e-002, max (m) = 2.550000e-001
    y-coordinate: min (m) = 0.000000e+000, max (m) = 2.500000e-002
  Volume statistics:
    minimum volume (m3): 2.463287e-009
    maximum volume (m3): 4.508038e-007
     total volume (m3): 4.190433e-004
    minimum 2d volume (m3): 3.000589e-007
    maximum 2d volume (m3): 3.019523e-006
  Face area statistics:
    minimum face area (m2): 4.199967e-004
    maximum face area (m2): 2.434403e-003
  Checking mesh.......................
 Done.
```

The mesh check report begins by listing the domain extents. The domain extents include the minimum and maximum x, y, and z coordinates in meters.

Then the volume statistics are provided, including the minimum, maximum, and total cell volume in m^3. A negative value for the minimum volume indicates that one or more cells have improper connectivity. Cells with a negative volume can often be identified using the **Iso-Value Adaption** dialog box to mark them for adaption and view them in the graphics window. For more information on creating and viewing isovalue adaption registers, see *Isovalue Adaption* (p. 1335). You *must* eliminate these negative volumes before continuing the flow solution process.

Next, the mesh report lists the face area statistics, including the minimum and maximum areas in m^2. A value of 0 for the minimum face area indicates that one or more cells have degenerated. As with negative volume cells, you *must* eliminate such faces. It is also recommended to correct cells that have nonzero face areas, if the values are very small.

Besides the information about the domain extents and statistics, the mesh check report will also display warnings based on the results of the checks previously described. You can specify that the mesh check report displays more detailed information about the various checks and the mesh failures, by using the following text command prior to performing the mesh check:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`mesh` → `check-verbosity`

You will then be prompted to enter the level of verbosity for the mesh check report. The possible levels include:

- `0`

  This is the default level, and only notifies you that checks are being performed (e.g., **Checking mesh...**). The report will look like the previous example; while warnings will be displayed below the domain extents and statistics, the names of the individual checks will not be listed as they are conducted.

- `1`

  This level lists the individual checks as they are performed (e.g., **Checking right-handed cells**). Any warnings that result will be displayed immediately below the check that produced it.

- `2`

  This level provides the maximum information about the mesh check. The report will list the individual checks as they are performed (e.g., **Checking right-handed cells**); any warnings that result will be displayed immediately below the check that produced it. Additional details about the check failure will also be displayed, such as the location of the problem or the affected cells.

6.5.2. Repairing Meshes

The mesh check report will indicate if the mesh has problems, such as left-handed faces and/or faces that have the wrong node order. You must take steps to repair such meshes. The simplest way to attempt to correct your mesh problems is to use the following text command:

`mesh` → `repair-improve` → `repair`

The `mesh/repair-improve/repair` text command will attempt to correct a number of problems identified by the mesh check, including cells that have:

- the wrong node order
- the wrong face handedness or that are not convex
- faces that are small or nonexistent
- very poor quality (see *Mesh Quality* (p. 144) for additional details)

Note that by default, the `repair` text command will only adjust the positions of interior nodes. If you want to also modify the nodes on the boundaries of the mesh, use the following text command *before* you repair the mesh:

`mesh` → `repair-improve` → `allow-repair-at-boundaries`

The `repair` text command may convert degenerate cells into polyhedra, based on skewness criteria (for more information on how cells are converted, see *Converting Skewed Cells to Polyhedra* (p. 182)). If you want to ensure that there are no polyhedra in the repaired mesh (e.g., if you if plan on performing mesh adaption), you must disable such conversions using the following text command *before* you repair the mesh:

`mesh` → `repair-improve` → `include-local-polyhedra-conversion-in-repair`

If you would like to only attempt to improve the poor quality cells, you can use the following text command:

`mesh` → `repair-improve` → `improve-quality`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the `improve-quality` text command multiple times, until the mesh is improved to your satisfaction. For greater control over the degree to which the mesh is improved, you can perform quality-based smoothing (as described in *Quality-Based Smoothing* (p. 1351)).

It should be noted that both the `repair` and the `improve-quality` text commands can be CPU intensive, if there are a large number of poor quality cells in the mesh. If you are not as concerned about cell quality, you can attempt to fix only the left-handed faces and faces with the wrong node order. Begin by repairing the node order with the following text command:

`mesh` → `repair-improve` → `repair-face-node-order`

Because the left-handed faces may be a result of improper face node order, the previous text command may resolve both issues at the same time. Be sure to perform another mesh check after entering the `repair-face-node-order` command, to see if the mesh has been fully repaired.

If at any point the mesh check reveals that the mesh contains left-handed faces without any node order issues, you can attempt to repair the face handedness by modifying the cell centroids with the following text command:

`mesh` → `repair-improve` → `repair-face-handedness`

Once again, perform a mesh check to see if the text command was successful. The `repair-face-handedness` text command is most effective for cells with high aspect ratios.

If the mesh check report includes a warning message such as

```
WARNING: node on face thread 2 has multiple shadows.
```

it indicates the existence of duplicate shadow nodes. This error occurs only in meshes with periodic-type walls. You can repair such a mesh using the following text command:

`mesh` → `repair-improve` → `repair-periodic`

If the interface is rotational periodic, you will be prompted for the rotation angle.

6.6. Reporting Mesh Statistics

There are several methods for reporting information about the mesh after it has been read into ANSYS FLUENT. You can report the amount of memory used by the current problem, the mesh size, and statistics about the mesh partitions. Zone-by-zone counts of cells and faces can also be reported.

Information about mesh statistics is provided in the following sections:

6.6.1. Mesh Size
6.6.2. Memory Usage
6.6.3. Mesh Zone Information
6.6.4. Partition Statistics

6.6.1. Mesh Size

You can print out the numbers of nodes, faces, cells, and partitions in the mesh by selecting the **Mesh/Info/Size** menu item.

Mesh → **Info** → **Size**

A partition is a piece of a mesh that has been segregated for parallel processing (see *Parallel Processing* (p. 1585)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A sample of the resulting output follows:

```
Mesh Size

Level     Cells       Faces       Nodes        Partitions
    0      7917       12247       4468                 1

2 cell zones, 11 face zones.
```

If you are interested in how the cells and faces are divided among the different zones, you can use the **Mesh/Info/Zones** menu item, as described in *Mesh Zone Information* (p. 177).

If you are using the density-based coupled explicit solver, the mesh information will be printed for each grid level. The grid levels result from creating coarse grid levels for the FAS multigrid convergence acceleration (see Full-Approximation Storage (FAS) Multigrid in the Theory Guide). A sample of the resulting output is shown below:

```
Mesh Size

Level   Cells    Faces    Nodes     Partitions
 0       7917    12247     4468              1
 1       1347     3658        0              1
 2        392     1217        0              1
 3        133      475        0              1
 4         50      197        0              1
 5         17       78        0              1
2 cell zones, 11 face zones.
```

6.6.2. Memory Usage

During an ANSYS FLUENT session you may want to check the amount of memory used and allocated in the present analysis. ANSYS FLUENT has a feature that will report the following information: the numbers of nodes, faces, cells, edges, and object pointers (generic pointers for various mesh and graphics utilities) that are used and allocated; the amount of array memory (scratch memory used for surfaces) used and allocated; and the amount of memory used by the solver process.

You can obtain this information by selecting the **Mesh/Info/Memory Usage** menu item.

Mesh → **Info** → **Memory Usage**

The memory information will be different for UNIX and Windows systems.

6.6.2.1. UNIX Systems

On UNIX systems, note the following definitions related to process memory information:

- Process static memory is essentially the size of the code itself.
- Process dynamic memory is the allocated heap memory used to store the mesh and solution variables.
- Process total memory is the sum of static and dynamic memory.

6.6.2.2. Windows Systems

On Windows systems, note the following definitions related to process memory information:

- Process physical memory is the allocated heap memory currently resident in RAM.
- Process virtual memory is the allocated heap memory currently swapped to the Windows system page file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Process total memory is the sum of physical and virtual memory.

Note the following:

- The memory information does not include the static (code) memory.
- In the serial version of ANSYS FLUENT, the heap memory value includes storage for the solver (mesh and solution variables), and Cortex (GUI and graphics memory), since Cortex and the solver are contained in the same process.
- In the parallel version, Cortex runs in its own process, so the heap memory value includes storage for the mesh and solution variables only.

On Windows systems, you can also get more information on the ANSYS FLUENT process (or processes) by using the Task Manager (see your Windows documentation for details). For the serial version, the process image name will be something like `fl1206s.exe`. For the parallel version, examples of process image names are as follows: `cx392.exe` (Cortex) and `fl1206.exe` (solver host).

6.6.3. Mesh Zone Information

You can print information in the console about the nodes, faces, and cells in each zone using the **Mesh/Info/Zones** menu item.

Mesh → **Info** → **Zones**

The mesh zone information includes the total number of nodes and, for each face and cell zone, the number of faces or cells, the cell (and, in 3D, face) type (triangular, quadrilateral, etc.), the boundary condition type, and the zone ID. Sample output is shown below:

```
Zone sizes on domain 1:
21280 hexahedral cells, zone 4.
  532 quadrilateral velocity-inlet faces, zone 1.
  532 quadrilateral pressure-outlet faces, zone 2.
  1040 quadrilateral symmetry faces, zone 3.
  1040 quadrilateral symmetry faces, zone 7.
 61708 quadrilateral interior faces, zone 5.
   1120 quadrilateral wall faces, zone 6.
23493 nodes.
```

6.6.4. Partition Statistics

You can print mesh partition statistics in the console by selecting the **Mesh/Info/Partitions** menu item.

Mesh → **Info** → **Partitions**

The statistics include the numbers of cells, faces, interfaces, and neighbors of each partition. See *Interpreting Partition Statistics* (p. 1622) for further details, including sample output.

6.7. Converting the Mesh to a Polyhedral Mesh

Since the ANSYS FLUENT solver is face based, it supports polyhedral cells. The advantages that polyhedral meshes have shown over some of the tetrahedral or hybrid meshes is the lower overall cell count, almost 3-5 times lower for unstructured meshes than the original cell count. Currently, there are two options in ANSYS FLUENT that allow you to convert your nonpolyhedral mesh to a polyhedral mesh:

- Converting the entire domain into polyhedral cells (applicable only for meshes that contain tetrahedral and/or wedge/prism cells).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Converting skewed tetrahedral cells to polyhedral cells.

Information about polyhedral mesh conversion is provided in the following sections:

6.7.1. Converting the Domain to a Polyhedra
6.7.2. Converting Skewed Cells to Polyhedra

6.7.1. Converting the Domain to a Polyhedra

Conversion of a mesh to polyhedra only applies to 3D meshes that contain tetrahedral and/or wedge/prism cells.

To begin the conversion process, ANSYS FLUENT automatically decomposes each non-hexahedral cell into multiple sub-volumes called "duals" (the shaded regions seen in the 2D example in *Figure 6.36* (p. 179)). Each dual is associated with one of the original nodes of the cell. These duals are then agglomerated into polyhedral cells around the original nodes. Thus, the collection of duals from all cells sharing a particular node makes up each polyhedral cell (see *Figure 6.37* (p. 179)). The node that is now within the polyhedral cell is no longer needed and is removed.

To better understand how duals are formed, you can consider the straightforward case of a tetrahedral mesh. Each of the cells are decomposed in the following manner: first, new edges are created on each face between the face centroid and the centroids of the edges of that face. Then, new faces are created within the cell by connecting the cell centroid to the new edges on each face. These interior faces establish the boundaries between the duals of a cell, and divide the cell into 4 sub-volumes. These dividing faces may be adjusted and merged with neighboring faces during the agglomeration process, in order to minimize the number of faces on the resultant polyhedral cell.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.36 Connection of Edge Centroids with Face Centroids

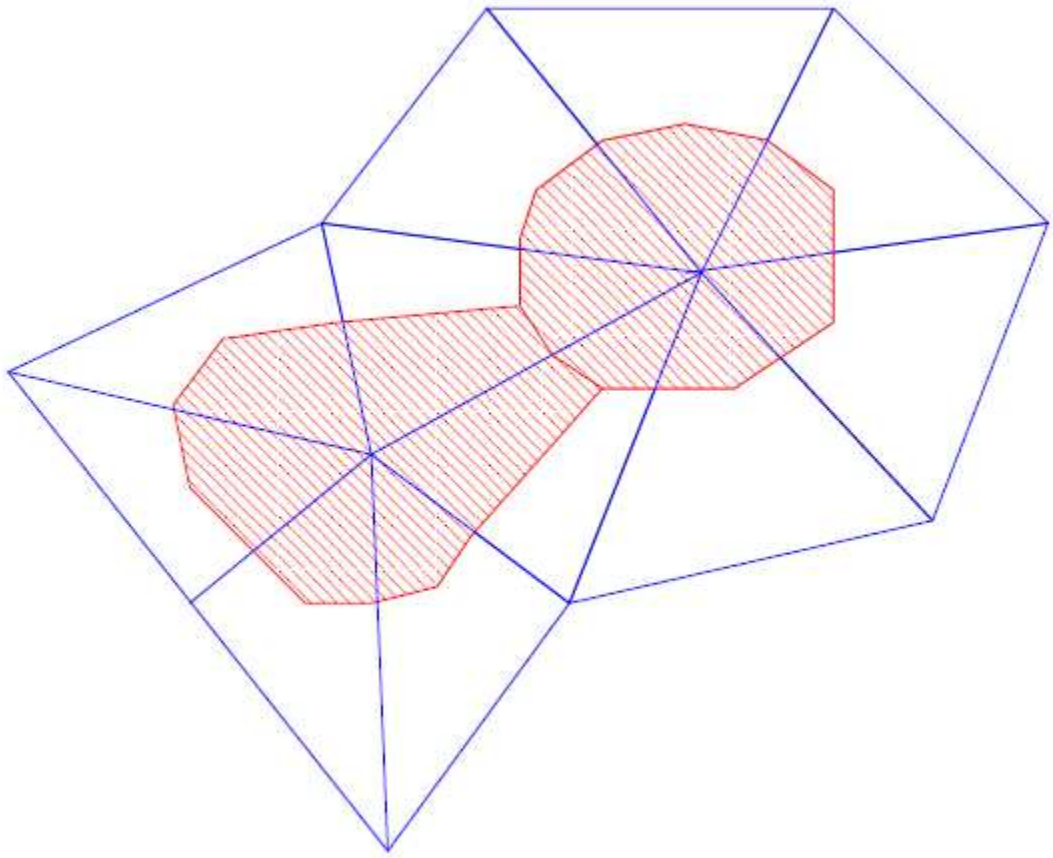
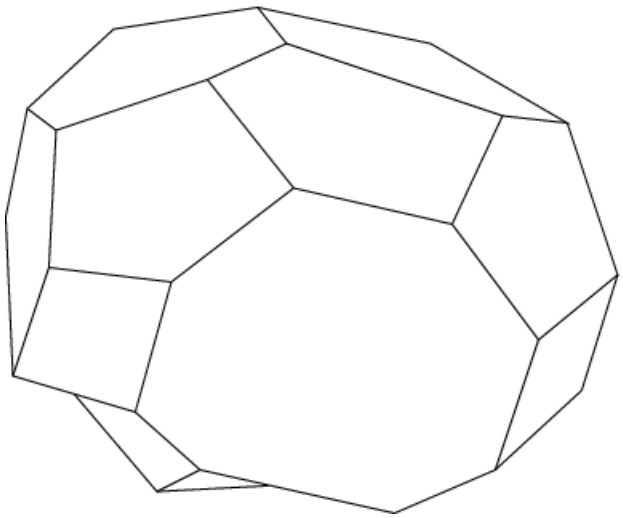


Figure 6.37 A Polyhedral Cell

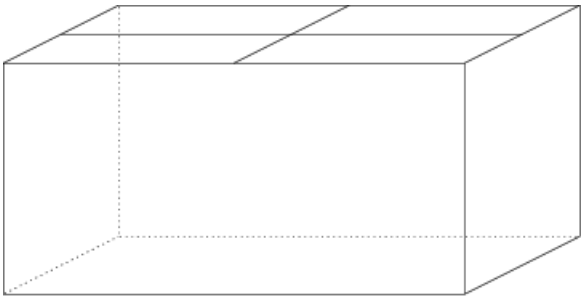


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

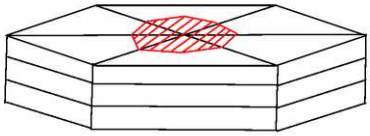
Hexahedral cells are not converted to polyhedra when the domain is converted, except when they border non-hexahedral cells. When the neighboring cell is reconfigured as polyhedra, the shared face of the hexahedral cell is decomposed into multiple faces as well, resulting in a polyhedral cell. In such a case the shape of the original hexahedral cell is preserved (i.e. the overall dimensions of the cell stay the same), but the converted cell has more than the original 6 faces (see *Figure 6.38* (p. 180)).

Figure 6.38 A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape



Conversion proceeds in a slightly different manner in boundary layers that are modeled using thin wedge/prism cells. These cells are decomposed in the plane of the boundary surface, but not in the direction normal to the surface. The resulting polyhedra will therefore preserve the thickness of the original wedge/prism cells (*Figure 6.39* (p. 180)). In most cases, the cell count in the new polyhedral boundary layer will be lower than the original boundary layer.

Figure 6.39 Treatment of Wedge Boundary Layers



To convert the entire domain of your mesh, use the **Mesh/Polyhedra/Convert Domain** menu.

Mesh → **Polyhedra** → **Convert Domain**

The following is an example of the resulting message printed in the console:

```
  Setup conversion to polyhedra.
  Converting domain to polyhedra...

       Creating polyhedra zones.
       Processing face zones...........
       Processing cell zones...
       Building polyhedra mesh..................
       Optimizing polyhedra mesh.....
 >> Reordering domain using Reverse Cuthill-McKee method:
     zones, cells, faces, done.
  Bandwidth reduction = 1796/247 = 7.27
  Done.
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.40 (p. 181), the original tetrahedral mesh of a section of a manifold, is compared to *Figure 6.41* (p. 181) which is the resulting mesh after the entire domain is converted to a polyhedra.

Figure 6.40 The Original Tetrahedral Mesh

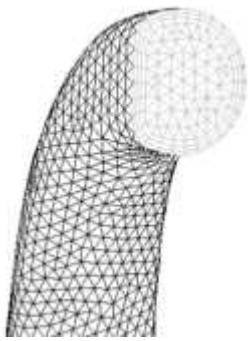


Figure 6.41 The Converted Polyhedral Mesh

6.7.1.1. Limitations

Some limitations you will find with polyhedral meshes that you generally do not experience with other cell types include:

- Meshes that already contain polyhedral cells cannot be converted.
- Meshes with hanging nodes will not be converted. This includes meshes that have undergone hanging node adaption (see Hanging Node Adaption in the Theory Guide), as well as meshes generated by the GAMBIT **Hex Core** meshing scheme or the TGrid **Hexcore** menu option.
- The following mesh manipulation tools are not available for polyhedral meshes:
  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - fuse
  - skewness smoothing
  - swapping will not affect polyhedral cells
- Meshes in which the domain has been converted to polyhedral cells are not eligible for adaption. For more information about adaption, see *Adapting the Mesh* (p. 1325).
- The dynamic mesh model cannot be used on polyhedral meshes. The dynamic mesh model will automatically be disabled if a case with the dynamic mesh option enabled is converted.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.2. Converting Skewed Cells to Polyhedra

Another method of cell agglomeration is the skewness-based cluster approach. This type of conversion is designed to convert only part of the domain. The objective is to convert only skewed tetrahedral cells above a specified cell equivolume skewness threshold into polyhedra. By converting the highly skewed tetrahedral cells, the quality of the mesh can be improved significantly.

A different algorithm is used for local conversion. This algorithm evaluates each highly skewed tetrahedral cell and all of the surrounding cells, to select an edge on the highly skewed cell that best matches criteria for cell agglomeration. Then all of the cells which share this edge are combined into a polyhedral cell. During the process, the data is interpolated from the original cells to the resultant polyhedra.

6.7.2.1. Limitations

There are certain limitations with this type of conversion:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - fuse
  - skewness smoothing
  - swapping will not affect polyhedral cells
- The polyhedral cells that result from the conversion are not eligible for adaption. For more information about adaption, see *Adapting the Mesh* (p. 1325).
- Only tetrahedral cells are converted, as all other cells are skipped.
- Meshes with hanging nodes will not be converted. This includes meshes that have undergone hanging node adaption (see Hanging Node Adaption in the Theory Guide), as well as meshes generated by the GAMBIT **Hex Core** meshing scheme or the TGrid **Hexcore** menu option (hexcore meshes). Note that if the mesh is a hexcore mesh in which the transitional cells have been converted to polyhedra, then it does not have hanging nodes and can therefore be converted.

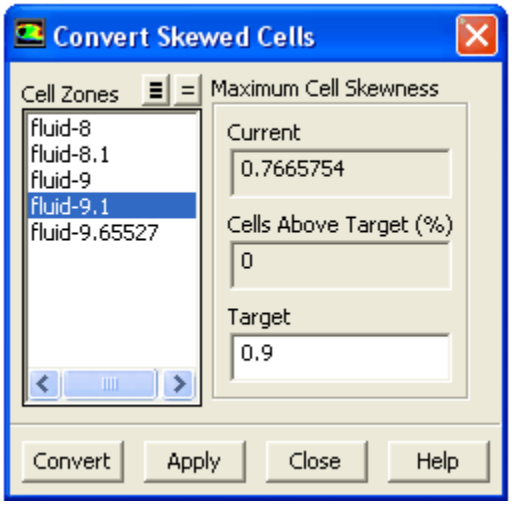
6.7.2.2. Using the Convert Skewed Cells Dialog Box

To convert skewed cells in your domain to polyhedral cells, go to the **Convert Skewed Cells** dialog box.

Mesh → **Polyhedra** → **Convert Skewed Cells...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.42 The Convert Skewed Cells Dialog Box



1. Select the zone(s) you want to consider for local polyhedra conversion from the **Cell Zones** selection list. After the zone selection is made, the **Current** value of the **Maximum Cell Skewness** and the percentage of **Cells Above Target** are displayed.
2. Specify the maximum allowable cell skewness in the **Target** text-entry box, and press `<Enter>` to update the **Cells Above Target**.

   Important

   The **Cells Above Target (%)** should be only a couple of percentage points, otherwise the conversion will be ineffective due to the high face count.

3. Click the **Convert** button.

The number of created polyhedra and the resulting maximum cell skewness will be printed in the console.

6.8. Modifying the Mesh

There are several ways in which you can modify or manipulate the mesh after it has been read into ANSYS FLUENT. You can scale or translate the mesh, merge or separate zones, create or slit periodic zones, and fuse boundaries. In addition, you can reorder the cells in the domain to decrease bandwidth. Smoothing and diagonal swapping, which can be used to improve the mesh, are described in *Improving the Mesh by Smoothing and Swapping* (p. 1349). Methods for partitioning meshes to be used in a parallel solver are discussed in *Mesh Partitioning and Load Balancing* (p. 1603).

Important

Whenever you modify the mesh, you should be sure to save a new case file (and a data file, if data exists). If you have old data files that you would like to be able to read in again, be sure to retain the original case file as well, as the data in the old data files may not correspond to the new case file.

Information about mesh manipulation is provided in the following sections:

6.8.1. Merging Zones

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8.2. Separating Zones
6.8.3. Fusing Face Zones
6.8.4. Creating Conformal Periodic Zones
6.8.5. Slitting Periodic Zones
6.8.6. Slitting Face Zones
6.8.7. Orienting Face Zones
6.8.8. Extruding Face Zones
6.8.9. Replacing, Deleting, Deactivating, and Activating Zones
6.8.. Replacing the Mesh
6.8.11. Reordering the Domain and Zones
6.8.12. Scaling the Mesh
6.8.13. Translating the Mesh
6.8.14. Rotating the Mesh

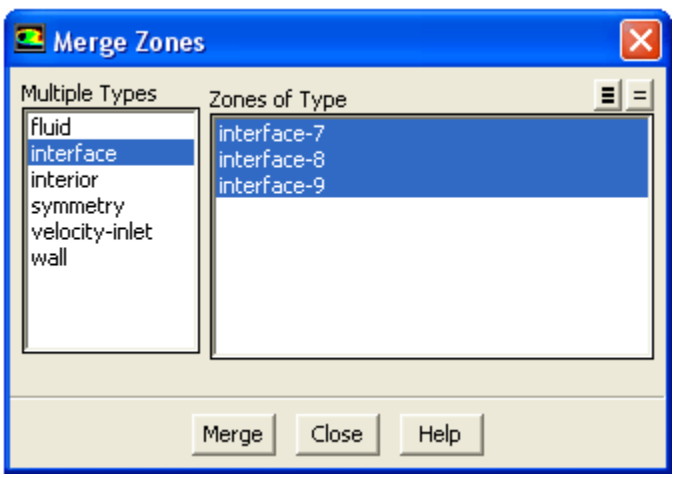
6.8.1. Merging Zones

To simplify the solution process, you may want to merge zones. Merging zones involves combining multiple zones of similar type into a single zone. Setting boundary conditions and postprocessing may be easier after you have merged similar zones.

Zone merging is performed in the *Merge Zones Dialog Box* (p. 2077) (*Figure 6.43* (p. 184)).

Mesh → Merge...

Figure 6.43 The Merge Zones Dialog Box



6.8.1.1. When to Merge Zones

ANSYS FLUENT allows you to merge zones of similar type into a single zone. This is not necessary unless the number of zones becomes prohibitive to efficient setup or postprocessing of the numerical analysis. For example, setting the same boundary condition parameters for a large number of zones can be time-consuming and may introduce inconsistencies. In addition, the postprocessing of the data often involves surfaces generated using the zones. A large number of zones often translates into a large number of surfaces that must be selected for the various display options, such as color contouring. Fortunately, surfaces can also be merged (see *Grouping, Renaming, and Deleting Surfaces* (p. 1376)), minimizing the negative impact of a large number of zones on postprocessing efficiency.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Although merging zones can be helpful, there may be cases where you will want to retain a larger number of zones. Since the merging process is not fully reversible, a larger number of zones provides more flexibility in imposing boundary conditions. Although a large number of zones can make selection of surfaces for display tedious, it can also provide more choices for rendering the mesh and the flow-field solution. For instance, it can be difficult to render an internal flow-field solution. If the outer domain is composed of several zones, the meshes of subsets of these zones can be plotted along with the solution to provide the relationship between the geometry and solution field. Merging zones may also adversely affect dynamic zones and mesh interfaces.

6.8.1.2. Using the Merge Zones Dialog Box

The procedure for merging multiple zones of the same type into a single zone is as follows:

1. Select the zone type in the **Multiple Types** list. This list contains all the zone types for which there are multiple zones. When you choose a type from this list, the corresponding zones will appear in the **Zones of Type** list.
2. Select two or more zones in the **Zones of Type** list.
3. Click the **Merge** button to merge the selected zones. Note that if your case file has dynamic zones or mesh interfaces, the **Warning** dialog box will open before the merge is initiated, allowing you to specify whether you want to delete such zones or interfaces first (see *Warning Dialog Box* (p. 2077) for details).

Important

Remember to save a new case file (and a data file, if data exists).

6.8.2. Separating Zones

Upon reading a mesh file, ANSYS FLUENT automatically performs zone separations in two conditions. If a face zone is attached to multiple cells zones in the preprocessor, the face zone will be separated so that each one is attached to only one cell zone. Furthermore, if you have defined an internal face as a wall type, an additional shadow wall zone will be generated (e.g., for a wall named **baffle**, a shadow wall zone named **baffle-shadow** will be generated).

There are several methods available in ANSYS FLUENT that allow you to manually separate a single face or cell zone into multiple zones of the same type. If your mesh contains a zone that you want to break up into smaller portions, you can make use of these features. For example, if you created a single wall zone when generating the mesh for a duct, but you want to specify different temperatures on specific portions of the wall, you will need to break that wall zone into two or more wall zones. If you plan to solve a problem using the sliding mesh model or multiple reference frames, but you forgot to create different fluid zones for the regions moving at different speeds, you will need to separate the fluid zone into two or more fluid zones.

Important

- After performing any of these separations, you should save a new case file. If data exists, it is automatically assigned to the proper zones when separation occurs, so you should also write a new data file. The old data cannot be read on top of the case file in which the zones have changed.
- The maximum number of zones into which you can separate any one face zone or cell zone is 32.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are four ways to separate face zones and two ways to separate cell zones. The face separation methods will be described first, followed by the cell separation tools. Slitting (decoupling) of periodic zones is discussed in *Slitting Periodic Zones* (p. 192).

Note that all of the separation methods allow you to report the result of the separation before you commit to performing it.

6.8.2.1. Separating Face Zones

6.8.2.1.1. Methods for Separating Face Zones

For geometries with sharp corners, it is often easy to separate face zones based on significant angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle will be placed in different zones. For example, if your mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, you would specify a significant angle of 89°. Since the normal vector for each cube side differs by 90° from the normals of its adjacent sides, each of the 6 sides will be placed in a different wall zone.

If you have a small face zone and would like to put each face in the zone into its own zone, you can do so by separating the faces based on face. Each individual face (triangle, quad, or polygon) will be separated into different zones.

You can also separate face zones based on the marks stored in adaption registers. For example, you can mark cells for adaption based on their location in the domain (region adaption), their boundary closeness (boundary adaption), isovalues of some variable, or any of the other adaption methods discussed in *Adapting the Mesh* (p. 1325). When you specify which register is to be used for the separation of the face zone, all faces of cells that are marked will be placed into a new face zone. (Use the *Manage Adaption Registers Dialog Box* (p. 2129) to determine the ID of the register you wish to use.)

Finally, you can separate face zones based on contiguous regions. For example, when you use coupled wall boundary conditions you need the faces on the zone to have a consistent orientation. Consistent orientation can only be guaranteed on contiguous regions, so you may need to separate face zones to allow proper boundary condition specification.

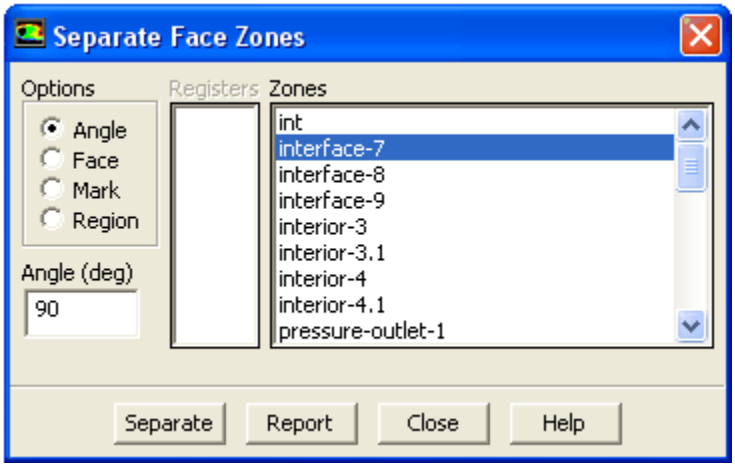
6.8.2.1.2. Inputs for Separating Face Zones

To break up a face zone based on angle, face, adaption mark, or region, use the *Separate Face Zones Dialog Box* (p. 2078) (*Figure 6.44* (p. 187)).

Mesh → **Separate** → **Faces...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.44 The Separate Face Zones Dialog Box



Important

If you are planning to separate face zones, you should do so before performing any adaptions using the (default) hanging node adaption method. Face zones that contain hanging nodes cannot be separated.

The steps for separating faces are as follows:

1. Select the separation method (**Angle**, **Face**, **Mark**, or **Region**) under **Options**.
2. Specify the face zone to be separated in the **Zones** list.
3. If you are separating by face or region, skip to the next step. Otherwise, do one of the following:
   - If you are separating faces by angle, specify the significant angle in the **Angle** field.
   - If you are separating faces by mark, select the adaption register to be used in the **Registers** list.
4. (optional) To check what the result of the separation will be before you actually separate the face zone, click the **Report** button. The report will look like the following example:

```
45 faces in contiguous region 0
30 faces in contiguous region 1
11 faces in contiguous region 2
14 faces in contiguous region 3
Separates zone 4 into 4 zone(s).
```

5. To separate the face zone, click the **Separate** button. A report will be printed in the console like the following example:

```
45 faces in contiguous region 0
30 faces in contiguous region 1
11 faces in contiguous region 2
14 faces in contiguous region 3
Separates zone 4 into 4 zone(s).
Updating new zone information ...
   created new zone wall-4:001 from wall-4
   created new zone wall-4:002 from wall-4
   created new zone wall-4:010 from wall-4
done.
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

When you separate the face zone by adaption mark, you may sometimes find that a face of a corner cell will be placed in the wrong face zone. You can usually correct this problem by performing an additional separation, based on angle, to move the offending face to a new zone. You can then merge this new zone with the zone in which you want the face to be placed, as described in *Merging Zones* (p. 184).

6.8.2.2. Separating Cell Zones

6.8.2.2.1. Methods for Separating Cell Zones

If you have two or more enclosed cell regions sharing internal boundaries (as shown in *Figure 6.45* (p. 188)), but all of the cells are contained in a single cell zone, you can separate the cells into distinct zones using the separation-by-region method. Note that if the shared internal boundary is of type **interior**, you must change it to another double-sided face zone type (**fan**, **radiator**, etc.) prior to performing the separation.

Figure 6.45 Cell Zone Separation Based on Region

You can also separate cell zones based on the marks stored in adaption registers. You can mark cells for adaption using any of the adaption methods discussed in *Adapting the Mesh* (p. 1325) (e.g., you can mark cells with a certain isovalue range or cells inside or outside a specified region). When you specify which register is to be used for the separation of the cell zone, cells that are marked will be placed into a new cell zone. (Use the *Manage Adaption Registers Dialog Box* (p. 2129) to determine the ID of the register you wish to use.)

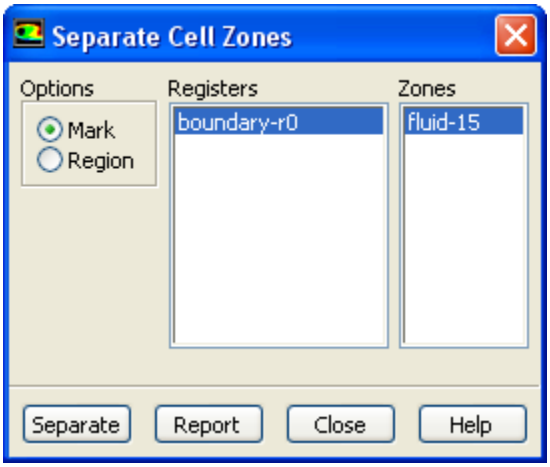
6.8.2.2.2. Inputs for Separating Cell Zones

To break up a cell zone based on region or adaption mark, use the *Separate Cell Zones Dialog Box* (p. 2080) (*Figure 6.46* (p. 189)).

Mesh → **Separate** → **Cells...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.46 The Separate Cell Zones Dialog Box



Important

If you are planning to separate cell zones, you should do so before performing any adaptions using the (default) hanging node adaption method. Cell zones that contain hanging nodes cannot be separated.

The steps for separating cells are as follows:

1. Select the separation method (**Mark** or **Region**) under **Options**.
2. Specify the cell zone to be separated in the **Zones** list.
3. If you are separating cells by mark, select the adaption register to be used in the **Registers** list.
4. (optional) To check what the result of the separation will be before you actually separate the cell zone, click the **Report** button. The report will look like this:

```
Separates zone 14 into two zones, with 1275 and 32 cells.
```

5. To separate the cell zone, click the **Separate** button. ANSYS FLUENT will print the following information:

```
Separates zone 14 into two zones, with 1275 and 32 cells.
No faces marked on thread, 2
No faces marked on thread, 3
No faces marked on thread, 1
No faces marked on thread, 5
No faces marked on thread, 7
No faces marked on thread, 8
No faces marked on thread, 9
No faces marked on thread, 61
Separates zone 62 into two zones, with 1763 and 58 faces.
All faces marked on thread, 4
No faces marked on thread, 66
Moved 32 cells from cell zone 14 to zone 10
Updating new zone information ...
    created new zone interior-4:010 from interior-4
    created new zone interior-6:009 from interior-6
    created new zone fluid-14:008 from fluid-14
done.
```

As shown in the example above, separation of a cell zone will often result in the separation of face zones as well. When you separate by mark, faces of cells that are moved to a new zone will be placed in a new

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

face zone. When you separate by region, faces of cells that are moved to a new zone will not necessarily be placed in a new face zone.

If you find that any faces are placed incorrectly, see the comment above, at the end of the inputs for face zone separation.

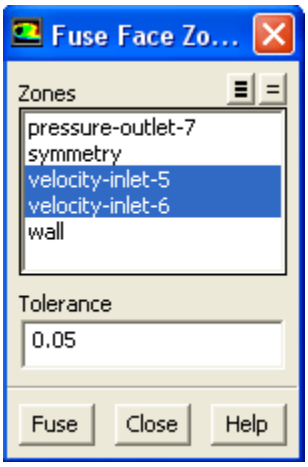
6.8.3. Fusing Face Zones

The face-fusing utility is a convenient feature that can be used to fuse boundaries (and merge duplicate nodes and faces) created by assembling multiple mesh regions. When the domain is divided into subdomains and the mesh is generated separately for each subdomain, you will combine the subdomains into a single file before reading the mesh into the solver. For details, see *Reading Multiple Mesh/Case/Data Files* (p. 157). This situation could arise if you generate each block of a multiblock mesh separately and save it to a separate mesh file. Another possible scenario is that you decided, during mesh generation, to save the mesh for each part of a complicated geometry as a separate part file. (Note that the mesh node locations need not be identical at the boundaries where two subdomains meet; see *Non-Conformal Meshes* (p. 161) for details.)

The **Fuse Face Zones** dialog box (*Figure 6.47* (p. 190)) allows you to merge the duplicate nodes and delete these artificial internal boundaries.

Mesh → **Fuse...**

Figure 6.47 The Fuse Face Zones Dialog Box



The boundaries on which the duplicate nodes lie are assigned zone ID numbers (just like any other boundary) when the mesh files are combined, as described in *Reading Multiple Mesh/Case/Data Files* (p. 157). You need to keep track of the zone ID numbers when `tmerge` or TGrid reports its progress or, after the complete mesh is read in, display all boundary mesh zones and use the mouse-probe button to determine the zone names (see *Controlling the Mouse Button Functions* (p. 1427) for information about the mouse button functions).

6.8.3.1. Inputs for Fusing Face Zones

The steps for fusing face zones are as follows:

1. Select the zones to be fused in the **Zones** list.
2. Click the **Fuse** button to fuse the selected zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If all of the appropriate faces do not get fused using the default **Tolerance**, you should increase it and attempt to fuse the zones again. (This tolerance is the same as the matching tolerance discussed in *Creating Conformal Periodic Zones* (p. 191).) The **Tolerance** should not exceed 0.5, or you may fuse the wrong nodes.

When fusing face zones using the GUI, ANSYS FLUENT automatically assigns a new name to the fused interface zone. If you would like to preserve the original name of one of the face zones being fused, you can use the `mesh/modify-zones/fuse-face-zones` text command, as shown in the following example.

```
/mesh/modify-zones > fuse-face-zones

()
Zone to fuse(1) [()]  top

Zone to fuse(2) [()]  bottom.1

Zone to fuse(3) [()]  <Enter>

 all 378 faces matched for zones 3 and 12.
   fusing created new thread, interior-18.
The fused zone name: (automatic bottom.1 top)
Enter name [automatic]  top

  Name of zone 18 is changed into top.

Fused list of zones.
```

Important

Remember to save a new case file (and a data file, if data exists) after fusing faces.

6.8.3.1.1. Fusing Zones on Branch Cuts

Meshes imported from structured mesh generators or solvers (such as FLUENT 4) can often be O-type or C-type meshes with a reentrant branch cut where coincident duplicate nodes lie on a periodic boundary. Since ANSYS FLUENT uses an unstructured mesh representation, there is no reason to retain this artificial internal boundary. (You can preserve these periodic boundaries and the solution algorithm will solve the problem with periodic boundary conditions.)

To fuse this periodic zone with itself, you must first slit the periodic zone, as described in *Slitting Periodic Zones* (p. 192). This will create two symmetry zones that you can fuse using the procedure above.

Note that if you need to fuse portions of a non-periodic zone with itself, you must use the `mesh/modify-zones/fuse-face-zones` text command.

`mesh` → `modify-zones` → `fuse-face-zones`

This command will prompt you for the name or ID of each zone to be fused. (You will enter the same zone twice.) To change the node tolerance, use the `mesh/modify-zones/matching-tolerance` text command.

6.8.4. Creating Conformal Periodic Zones

ANSYS FLUENT allows you to set up periodic boundaries using either conformal or non-conformal periodic zones. You can assign periodicity to your mesh by coupling a pair of face zones. If the two zones have identical node and face distributions, you can create a conformal periodic zone. If the two zones are not identical at the boundaries within a specified tolerance, then you can create a periodic zone at a non-conformal mesh interface (*Non-Conformal Meshes* (p. 161)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Remember to save a new case file (and a data file, if data exists) after creating or slitting a periodic boundary.

To create conformal periodic boundaries, you will use the `mesh/modify-zones/make-periodic` text command.

`mesh` → `modify-zones` → `make-periodic`

You will need to specify the two face zones that will comprise the periodic pair (you can enter their full names or just their IDs), and indicate whether they are rotationally or translationally periodic. The order in which you specify the periodic zone and its matching shadow zone is not significant.

```
  /mesh/modify-zones>  make-periodic

 Periodic zone [()] 1
 Shadow zone [()] 4
 Rotational periodic? (if no, translational) [yes] n
 Create periodic zones? [yes] yes
 Auto detect translation vector? [yes] yes

     computed translation deltas: -2.000000 -2.000000
     all 10 faces matched for zones 1 and 4.

     zone 4 deleted

     created periodic zones.
```

When you create a conformal periodic boundary, the solver will check to see if the faces on the selected zones "match" (i.e., whether or not the nodes on corresponding faces are coincident). The matching tolerance for a face is a fraction of the minimum edge length of the face. If the periodic boundary creation fails, you can change the matching tolerance using the `mesh/modify-zones/matching-tolerance` text command, but it should not exceed 0.5 or you may match up the periodic zones incorrectly and corrupt the mesh.

`mesh` → `modify-zones` → `matching-tolerance`

For information about creating non-conformal periodic boundaries, see *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169).

6.8.5. Slitting Periodic Zones

If you wish to decouple the zones in a periodic pair, you can use the `mesh/modify-zones/slit-periodic` text command.

`mesh` → `modify-zones` → `slit-periodic`

You will specify the periodic zone's name or ID, and the solver will decouple the two zones in the pair (the periodic zone and its shadow) and change them to two symmetry zones:

```
  /mesh/modify-zones> slit-periodic

 Periodic zone [()] periodic-1
 Slit periodic zone? [yes] yes

   Slit periodic zone.
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8.6. Slitting Face Zones

The face-zone slitting feature has two uses:

- You can slit a single boundary zone of any double-sided type (i.e., any boundary zone that has cells on both sides of it) into two distinct zones.
- You can slit a coupled wall zone into two distinct, uncoupled wall zones.

When you slit a face zone, the solver will duplicate all faces and nodes, except those nodes that are located at the ends (2D) or edges (3D) of the zone. One set of nodes and faces will belong to one of the resulting boundary zones and the other set will belong to the other zone. The only ill effect of the shared nodes at each end is that you may see some inaccuracies at those points when you graphically display solution data with a slit boundary. (Note that if you adapt the boundary after slitting, you will not be able to fuse the boundaries back together again.)

Before you can slit, you first need to select the zone in the **Boundary Condition** task page and change the **Type** to **wall**. Upon changing the **Type** to **wall**, another shadow zone will be created (i.e., if the original zone is called **rad-outlet**, another zone called **rad-outlet-shadow** will be created). Then you can apply the `mesh/modify-zones/slit-face-zone` text command on either of the walls (i.e., **rad-outlet** or **rad-outlet-shadow**) to separate them into two distinct walls.

For example, the outlet of the radiator in an underhood application is typically of **interior** type (i.e., has cells on both sides). If you know the mass flow rate through this zone (either from other CFD models or from test data) and want to apply it as a boundary condition at the radiator outlet, you first need to slit the radiator outlet. To be able to slit it, select **wall** from the **Type** drop-down list in the **Boundary Conditions** task page for this outlet. It will create a wall and a shadow. Then you can use the TUI command (`mesh/modify-zones/slit-face-zone`) to slit them. After it is slit, two additional walls will be created, one facing one side of the outlet and another facing the other. Then you can select the appropriate wall and change the **Type** to **mass-flow-inlet** and specify the mass flow rate using the *Mass-Flow Inlet Dialog Box* (p. 1832). There is no option of **mass-flow-inlet** without first slitting it.

Important

You should not confuse "slitting" a face zone with "separating" a face zone. When you slit a face zone, additional faces and nodes are created and placed in a new zone. When you separate a face zone, a new zone will be created, but no new faces or nodes are created; the existing faces and nodes are simply redistributed among the zones.

6.8.6.1. Inputs for Slitting Face Zones

If you wish to slit a face zone, you can use the `mesh/modify-zones/slit-face-zone` text command.

`mesh` → `modify-zones` → `slit-face-zone`

You will specify the face zone's name or ID, and the solver will replace the zone with two zones:

```
/mesh/modify-zones > slit-face-zone

 Face zone id/name [] wall-4

   zone 4 deleted
   face zone 4 created
   face zone 10 created
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Remember to save a new case file (and a data file, if data exists) after slitting faces.

6.8.7. Orienting Face Zones

The faces of a boundary face zone (which have cells only on one side) are oriented such that the normals are all pointing in one direction. However, for internal face zones (which have cells on both sides), the normals are allowed to point in either direction. To orient them so that they all point in one direction, you can use the following TUI command:

`mesh` → `modify-zones` → `orient-face-zone`

Having all of the normals oriented in one direction is needed for some boundary condition types. For example, the **fan** boundary condition type determines the flow direction based on its normals. If some of the normals are pointing in one direction and some in the other, the correct flow orientation cannot be determined, which leads to incorrect results.

6.8.8. Extruding Face Zones

The ability to extrude a boundary face zone allows you to extend the solution domain without having to exit the solver. A typical application of the extrusion capability is to extend the solution domain when recirculating flow is impinging on a flow outlet. The current extrusion capability creates prismatic or hexahedral layers based on the shape of the face and normal vectors computed by averaging the face normals to the face zone's nodes. You can define the extrusion process by specifying a list of displacements (in SI units) or by specifying a total distance (in SI units) and parametric coordinates.

Important

- Note that extrusion is not possible from boundary face zones that have hanging nodes.
- Extruding face zones is only allowed in the 3D version of ANSYS FLUENT.

6.8.8.1. Specifying Extrusion by Displacement Distances

You can specify the extrusion by entering a list of displacement distances (in SI units) using the `mesh/modify-zones/extrude-face-zone-delta` text command.

`mesh` → `modify-zones` → `extrude-face-zone-delta`

You will be prompted for the boundary face zone ID or name and a list of displacement distances.

6.8.8.2. Specifying Extrusion by Parametric Coordinates

You can specify the extrusion by specifying a distance (in SI units) and parametric coordinates using the `mesh/modify-zones/extrude-face-zone-para` text command.

`mesh` → `modify-zones` → `extrude-face-zone-para`

You will be prompted for the boundary face zone ID or name, a total distance, and a list of parametric coordinates. The list of parametric coordinates should begin with 0.0 and end with 1.0. For example, the following list of parametric coordinates would create two equally spaced extrusion layers: 0.0, 0.5, 1.0.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8.9. Replacing, Deleting, Deactivating, and Activating Zones

ANSYS FLUENT allows you to append or replace an existing cell zone in the mesh. You can also permanently delete a cell zone and all associated face zones, or temporarily deactivate and activate zones from your ANSYS FLUENT case.

6.8.9.1. Replacing Zones

This feature allows you to replace a small region of a computational domain with a new region of different mesh quality. This functionality will be required where you may want to make changes to the geometry or mesh quality for any part of the domain. This ability of ANSYS FLUENT will save you time, since you can modify only the required part of the domain without remeshing the whole domain every time.

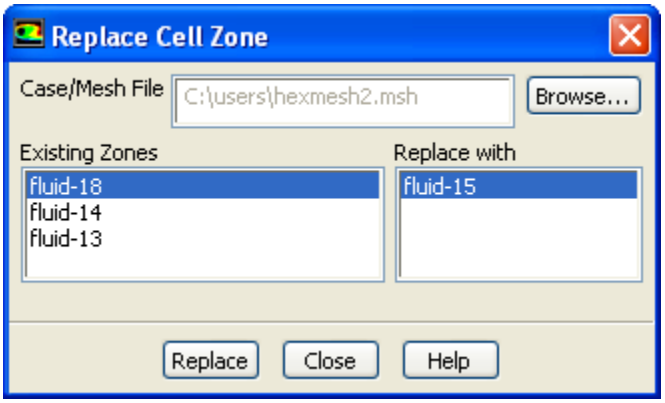
The replacement mesh must be prepared in advance, with the following considerations:

- The replacement mesh must contain a single cell zone.
- The cell zone in the replacement mesh must be of the same zone type as the zone that is being replaced.
- If the boundaries of the replacement mesh will form a non-conformal interface with existing zones (i.e., zones that are not replaced in the existing mesh), these boundaries should be based on the same geometry as the existing zones. If there are sharp features (e.g., 90-degree angles) or curvature in the mesh, it is especially important that both sides of the interface closely follow that feature.
- The replacement cell zone does not need to have the same name as the existing zone being replaced.
- When naming the boundary zones of the replacement cell zone, note that boundary conditions will be automatically transferred from the existing cell zone based on name. All boundaries of the replacement cell zone that have unique names will be assigned the default boundary condition for that zone type.

Replacing a zone is performed using the *Replace Cell Zone Dialog Box* (p. 2081) (*Figure 6.48* (p. 195)).

Mesh → **Zone** → **Replace...**

Figure 6.48 The Replace Cell Zone Dialog Box



To replace a zone, do the following:

1. Click **Browse...** and select the new or modified mesh containing the cell zone that will replace one of the cell zones in the current mesh.

   The name of the cell zone in the replacement mesh will be displayed in the ***Replace with*** *list.*

2. Under **Existing Zones**, select the zone you want to replace.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

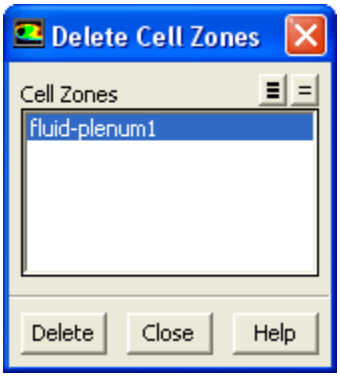
3. Under **Replace with**, select the zone from the replacement mesh.
4. Click **Replace** to replace the selected zone.
5. Set up the boundary conditions for any of the boundaries of the replacement mesh that do not share a name with one of the existing boundaries.

6.8.9.2. Deleting Zones

To permanently delete zones, select them in the *Delete Cell Zones Dialog Box* (p. 2082) (*Figure 6.49* (p. 196)), and click **Delete**.

Mesh → **Zone** → **Delete...**

Figure 6.49 The Delete Cell Zones Dialog Box



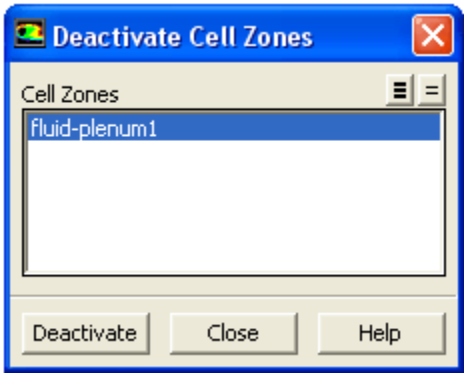
All of the cells, faces, and nodes associated with the cell zone will be deleted. If one of the faces is of type **interior** and borders another cell zone, the face will automatically be changed to a wall and will stay attached to the remaining cell zone.

6.8.9.3. Deactivating Zones

To deactivate zones, select them in the *Deactivate Cell Zones Dialog Box* (p. 2083) (*Figure 6.50* (p. 196)), and click **Deactivate**.

Mesh → **Zone** → **Deactivate...**

Figure 6.50 The Deactivate Cell Zones Dialog Box



Deactivation will separate all relevant interior face zones (i.e., fan, interior, porous-jump, or radiator) into wall and wall-shadow pairs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

When you deactivate a zone using the **Deactivate Cell Zones** dialog box, ANSYS FLUENT will remove the zone from the mesh and from all relevant solver loops.

To deactivate selected cell zones in parallel

1. Read in the data file, or initialize your solution.
2. Select the zone(s) to be deactivated under **Cell Zones**.
3. Click the **Deactivate** button.

Important

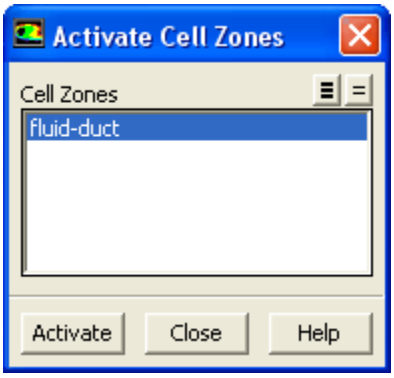
If you have neither read in the data file nor initialized the solution prior to clicking the **Deactivate** button, then the selected cell zones will only be *marked* for deactivation. The zones will not be deactivated until data is read or the solution is initialized.

6.8.9.4. Activating Zones

You can reactivate the zones and recover the last data available for them using the *Activate Cell Zones Dialog Box* (p. 2083) (*Figure 6.51* (p. 197)).

Mesh → **Zone** → **Activate...**

Figure 6.51 The Activate Cell Zones Dialog Box



Note

The **Activate Cell Zones** dialog box will only be populated with zones that were previously deactivated.

After reactivation, you need to make sure that the boundary conditions for the wall and wall-shadow pairs are restored correctly to what you assigned before deactivating the zones. If you plan to reactivate them at a later time, make sure that the face zones that are separated during deactivation are not modified. Adaption, however, is supported.

To activate selected cell zones in parallel

1. Read in the data file, or initialize your solution.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the zone(s) to be activated under **Cell Zones**.
3. Click the **Activate** button.

Important

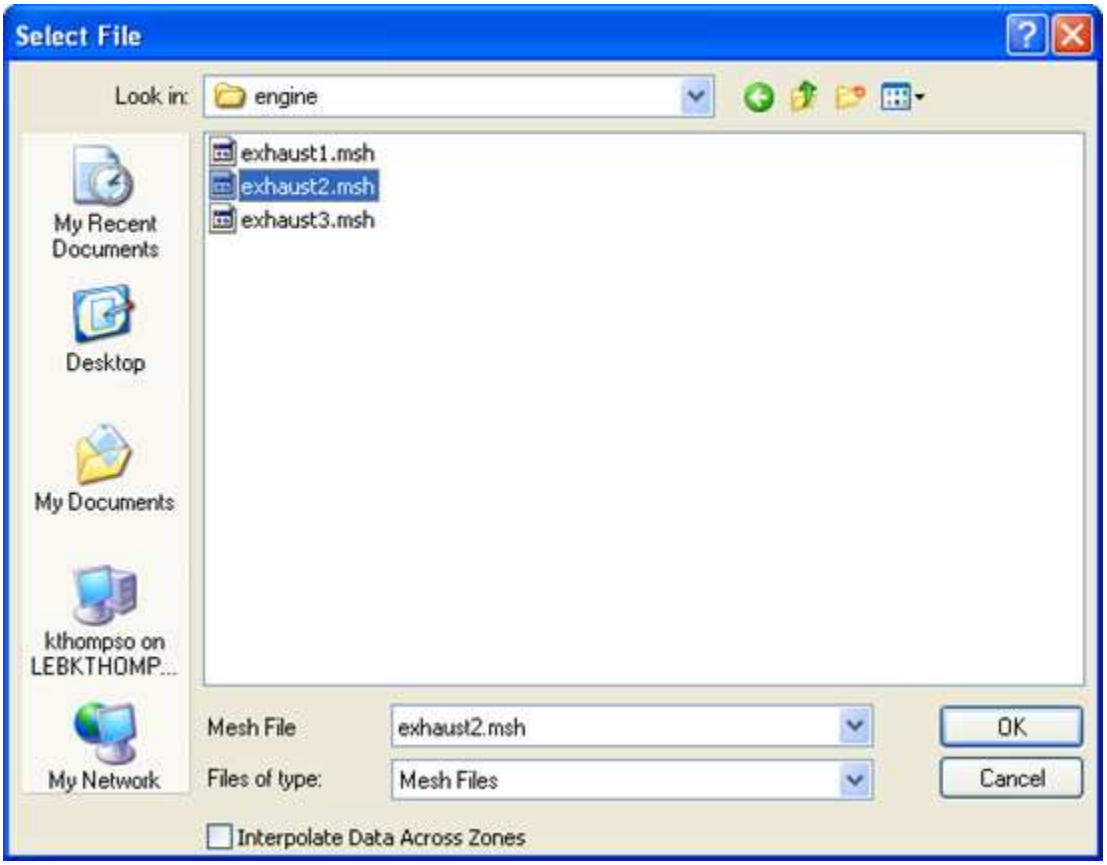
If you have neither read in the data file nor initialized the solution, prior to clicking the **Activate** button, then the selected cell zones will only be *marked* for activation. The zones will not be activated until data is read or the solution is initialized.

6.8.. Replacing the Mesh

ANSYS FLUENT allows you to replace your mesh, so that you can continue to run your simulation on a new mesh without having to manually copy the case file settings or interpolate the existing data. Replacing the mesh globally may be helpful, for example, if you would like to perform a mesh refinement study with a prepared series of meshes, or if your results indicate that your current mesh is not of sufficient quality. Note that the data interpolation is done automatically during the replacement if data exists (i.e., you have initialized the flow field or run a calculation).

Global mesh replacement is performed using the **Mesh/Replace...** menu item, which opens the **Select File** dialog box (*Figure 6.52* (p. 198)).

Figure 6.52 The Select File Dialog Box



6.8..1. Inputs for Replacing the Mesh

The steps for replacing the mesh using the **Select File** dialog box (*Figure 6.52* (p. 198)) are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the replacement mesh from the appropriate folder.
2. By default, data is interpolated between matching zone pairs (i.e., between the zones with the same names in both the current mesh and the replacement mesh). You can enable the **Interpolate Data Across Zones** option if you want to interpolate data across cell zones when replacing the mesh. This option is appropriate when the matching zone pairs do not have the same interior zone boundaries. Note that global conservation of data is not enforced when the **Interpolate Data Across Zones** option is enabled, so it should only be used when absolutely necessary. For best data interpolation, the zone boundaries of the replacement mesh should be coincident with those of the current mesh.
3. Click the **OK** button to replace the mesh.

6.8..2. Limitations

The following limitations apply when replacing meshes:

- The boundary conditions of the zones in the current mesh are mapped onto the zones with the same names in the replacement mesh, as described in *Reading and Writing Boundary Conditions* (p. 72). If your replacement mesh contains zones for which no match is found, these zones will have the default boundary conditions for that zone type.
- The replacement mesh is expected to be an ANSYS FLUENT mesh (`.msh`) file. You can select an ANSYS FLUENT case (`.cas`) file instead, but be aware that only the mesh information will be used and all of the setup information associated with that case file will be ignored.
- You should ensure that the replacement mesh has the same mesh scaling as your current mesh. The data interpolation will not work properly if the meshes are scaled differently.

6.8.11. Reordering the Domain and Zones

Reordering the domain can improve the computational performance of the solver by rearranging the nodes, faces, and cells in memory. The **Mesh/Reorder** submenu contains commands for reordering the domain and zones, and also for printing the bandwidth of the present mesh partitions. The domain can be reordered to increase memory access efficiency, and the zones can be reordered for user-interface convenience. The bandwidth provides insight into the distribution of the cells in the zones and in memory.

To reorder the domain, select the **Domain** menu item.

Mesh → **Reorder** → **Domain**

To reorder the zones, select the **Zones** menu item.

Mesh → **Reorder** → **Zones**

Finally, you can print the bandwidth of the present mesh partitions by selecting the **Print Bandwidth** menu item. This command prints the semi-bandwidth and maximum cell distance for each mesh partition.

Mesh → **Reorder** → **Print Bandwidth**

Important

Remember to save a new case file (and a data file, if data exist) after reordering.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8.11.1. About Reordering

The Reverse Cuthill-McKee algorithm [18] (p. 2209) is used in the reordering process to construct a "level tree" initiated from a "seed cell" in the domain. First a cell (called the seed cell) is selected using the algorithm of Gibbs, Poole, and Stockmeyer [26] (p. 2210). Each cell is then assigned to a level based on its distance from the seed cell. These level assignments form the level tree. In general, the faces and cells are reordered so that neighboring cells are near each other in the zone and in memory. Since most of the computational loops are over faces, you would like the two cells in memory cache at the same time to reduce cache and/or disk swapping—i.e., you want the cells near each other in memory to reduce the cost of memory access. The present scheme reorders the faces and cells in the zone, and the nodes, faces, and cells in memory.

You may also choose to reorder the zones. The zones are reordered by first sorting on zone type and then on zone ID. Zone reordering is performed simply for user-interface convenience.

A typical output produced using the domain reordering is shown below:

```
>> Reordering domain using Reverse Cuthill-McKee method:
       zones, cells, faces, done.
   Bandwidth reduction = 809/21 = 38.52
   Done.
```

If you print the bandwidth, you will see a report similar to the following:

```
Maximum cell distance = 21
```

The bandwidth is the maximum difference between neighboring cells in the zone—i.e., if you numbered each cell in the zone list sequentially and compared the differences between these indices.

6.8.12. Scaling the Mesh

Internally, ANSYS FLUENT stores the computational mesh in meters, the SI unit of length. When mesh information is read into the solver, it is assumed that the mesh was generated in units of meters. If your mesh was created using a different unit of length (inches, feet, centimeters, etc.), you must scale the mesh to meters. To do this, you can select from a list of common units to convert the mesh or you can supply your own custom scale factors. Each node coordinate will be multiplied by the corresponding scale factor.

Scaling can also be used to change the physical size of the mesh. For instance, you could stretch the mesh in the x direction by assigning a scale factor of 2 in the x direction and 1 in the y and z directions. This would double the extent of the mesh in the x direction. However, you should use anisotropic scaling with caution, since it will change the aspect ratios of the cells in your mesh.

Important

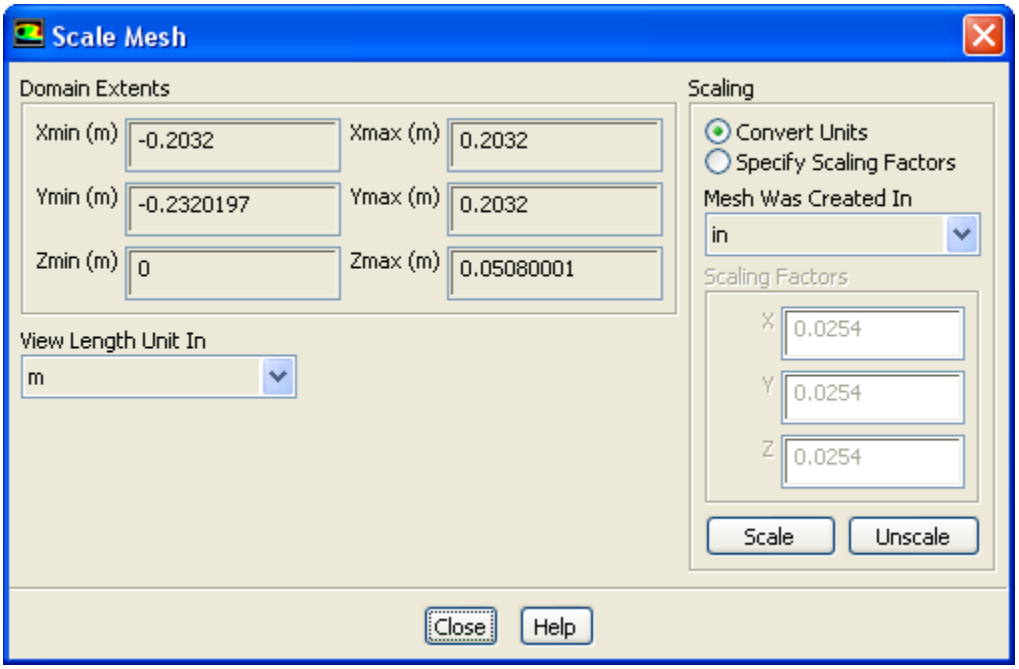
- If you plan to scale the mesh in any way, you should do so before you initialize the flow or begin calculations. Any data that exists when you scale the mesh will be invalid.
- It is a good practice to scale the mesh before setting up the case, especially when you plan to create mesh interfaces or shell conduction zones.

You will use the *Scale Mesh Dialog Box* (p. 1634) (*Figure 6.53* (p. 201)) to scale the mesh from a standard unit of measurement or to apply custom scaling factors.

General → **Scale...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.53 The Scale Mesh Dialog Box



6.8.12.1. Using the Scale Mesh Dialog Box

The procedure for scaling the mesh is as follows:

1. Use the conversion factors provided by ANSYS FLUENT by selecting **Convert Units** in the **Scaling** group box. Then indicate the units used when creating the mesh by selecting the appropriate abbreviation for meters, centimeters, millimeters, inches, or feet from the **Mesh Was Created In** drop-down list. The **Scaling Factors** will automatically be set to the correct values (e.g., 0.0254 meters/inch).

   If you created your mesh using units other than those in the **Mesh Was Created In** drop-down list, you can select **Specify Scaling Factors** and enter values for **X**, **Y**, and **Z** manually in the **Scaling Factors** group box (e.g., the number of meters per yard).

2. Click the **Scale** button. The **Domain Extents** will be updated to show the correct range in meters. If you prefer to use your original unit of length during the ANSYS FLUENT session, you can follow the procedure described below to change the unit.

6.8.12.1.1. Changing the Unit of Length

As mentioned in Step 2. of the previous section, when you scale the mesh you do not change the units; you just convert the original dimensions of your mesh points from your original units to meters by multiplying each node coordinate by the specified **Scaling Factors**. If you want to work in your original units, instead of in meters, you can make a selection from the **View Length Unit In** drop-down list. This updates the **Domain Extents** to show the range in your original units and automatically changes the length unit in the *Set Units Dialog Box* (p. 1637) (see *Customizing Units* (p. 126)). Note that this unit will be used for all future inputs of length quantities.

6.8.12.1.2. Unscaling the Mesh

If you use the wrong scale factor, accidentally click the **Scale** button twice, or wish to undo the scaling for any other reason, you can click the **Unscale** button. "Unscaling" simply divides each of the node coordinates

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by the specified **Scale Factors**. (Selecting **m** in the **Mesh Was Created In** list and clicking on **Scale** will *not* unscale the mesh.)

6.8.12.1.3. Changing the Physical Size of the Mesh

You can also use the *Scale Mesh Dialog Box* (p. 1634) to change the physical size of the mesh. For example, if your 2D mesh is 5 feet by 8 feet, and you want to model the same geometry with dimensions twice as big (10 feet by 16 feet), you can enter 2 for **X** and **Y** in the **Scaling Factors** group box and click **Scale**. The **Domain Extents** will be updated to show the new range.

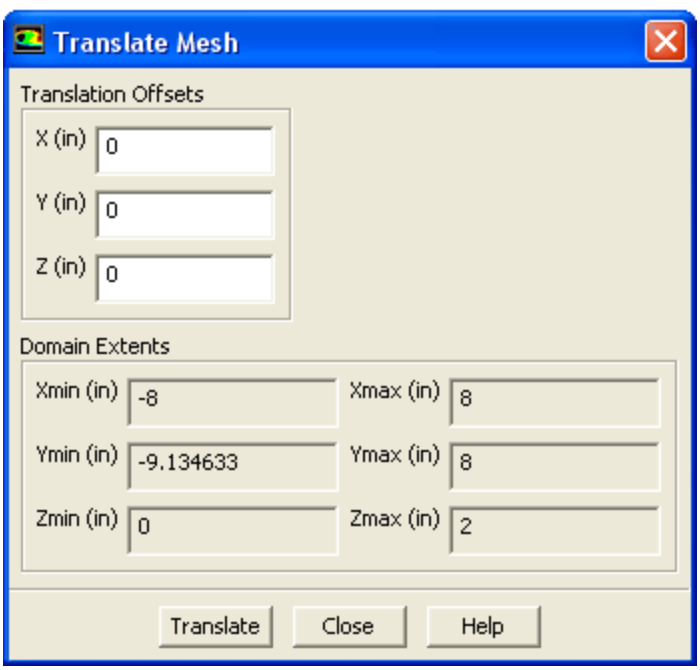
6.8.13. Translating the Mesh

You can "move" the mesh by applying prescribed offsets to the Cartesian coordinates of all the nodes in the mesh. This would be necessary for a rotating problem if the mesh were set up with the axis of rotation not passing through the origin, or for an axisymmetric problem if the mesh were set up with the axis of rotation not coinciding with the x axis. It is also useful if, for example, you want to move the origin to a particular point on an object (such as the leading edge of a flat plate) to make an XY plot have the desired distances on the x axis.

You can translate mesh points in ANSYS FLUENT using the *Translate Mesh Dialog Box* (p. 2084) (*Figure 6.54* (p. 202)).

Mesh $\rightarrow$ **Translate...**

Figure 6.54 The Translate Mesh Dialog Box



6.8.13.1. Using the Translate Mesh Dialog Box

The procedure for translating the mesh is as follows:

1. Enter the desired translations in the x, y, and (for 3D) z directions (i.e., the desired delta in the axes) in the **X**, **Y**, and **Z** text-entry boxes in the **Translation Offsets** group box. You can specify positive or negative real numbers in the current unit of length.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **Translate** button and redisplay the mesh. The **Domain Extents** will be updated to display the new extents of the translated mesh. (Note that the **Domain Extents** are purely informational; you cannot edit them manually.)

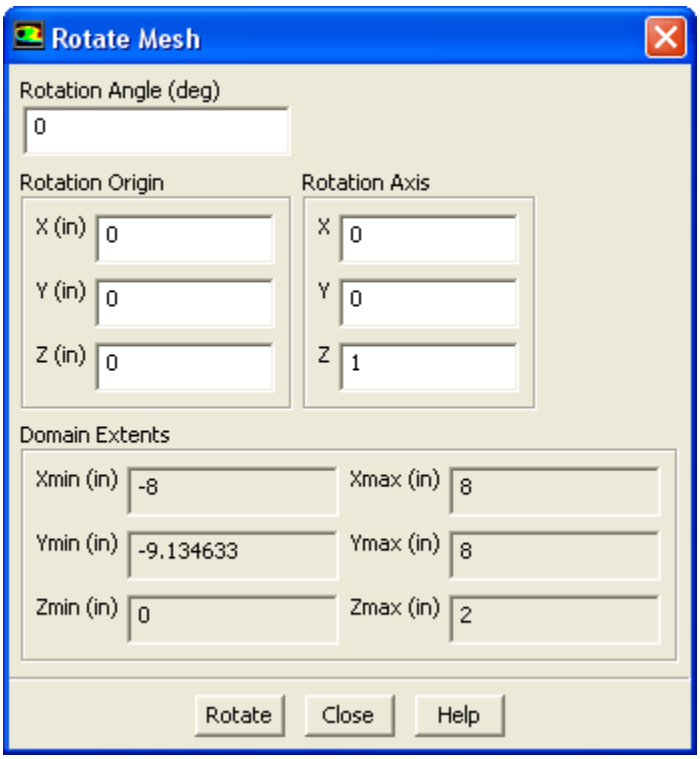
6.8.14. Rotating the Mesh

The ability to rotate the mesh is analogous to the ability to translate the mesh in ANSYS FLUENT. You can rotate the mesh about the x, y, or (for 3D) z axis and also specify the rotation origin. This option is useful in the cases where the structural mesh and the CFD mesh are offset by a small angle.

You can rotate the mesh in ANSYS FLUENT using the *Rotate Mesh Dialog Box* (p. 2085) (*Figure 6.55* (p. 203)).

Mesh → **Rotate...**

Figure 6.55 The Rotate Mesh Dialog Box



6.8.14.1. Using the Rotate Mesh Dialog Box

The procedure for rotating the mesh is as follows:

1. Specify the required **Rotation Angle** for the mesh. You can specify any positive or negative real number in the correct unit of angle.
2. In the **Rotation Origin** group box, enter **X**, **Y**, and (for 3D) **Z** coordinates to specify a new origin for the axis of rotation.
3. In the **Rotation Axis** group box, enter values for the **X**, **Y**, and (for 3D) **Z** axes to specify the vector for the axis of rotation.
4. Click the **Rotate** button and redisplay the mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Domain Extents will be updated to display the new extents of the rotated mesh. (Note that the Domain Extents are purely informational; you cannot edit them manually.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Cell Zone and Boundary Conditions

This chapter describes the cell zone and boundary condition options available in ANSYS FLUENT. Details regarding the cell zone and boundary condition inputs and the internal treatment at boundaries are provided.

The information in this chapter is divided into the following sections:

7.1. Overview
7.2. Cell Zone Conditions
7.3. Boundary Conditions
7.4. Non-Reflecting Boundary Conditions
7.5. User-Defined Fan Model
7.6. Profiles
7.7. Coupling Boundary Conditions with GT-Power
7.8. Coupling Boundary Conditions with WAVE

7.1. Overview

Cell zone and boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your ANSYS FLUENT simulations and it is important that they are specified appropriately.

7.1.1. Available Cell Zone and Boundary Types
7.1.2. The Cell Zone and Boundary Conditions Task Page
7.1.3. Changing Cell and Boundary Zone Types
7.1.4. Setting Cell Zone and Boundary Conditions
7.1.5. Copying Cell Zone and Boundary Conditions
7.1.6. Changing Cell or Boundary Zone Names
7.1.7. Defining Non-Uniform Cell Zones and Boundary Conditions
7.1.8. Defining and Viewing Parameters
7.1.9. Selecting Cell or Boundary Zones in the Graphics Display
7.1.10. Operating and Periodic Conditions
7.1.11. Highlighting Selected Boundary Zones
7.1.12. Saving and Reusing Cell Zone and Boundary Conditions

7.1.1. Available Cell Zone and Boundary Types

The boundary types available in ANSYS FLUENT are classified as follows:

- Flow inlet and exit boundaries: pressure inlet, velocity inlet, mass flow inlet, and inlet vent, intake fan, pressure outlet, pressure far-field, outflow, outlet vent, and exhaust fan.
- Wall, repeating, and pole boundaries: wall, symmetry, periodic, and axis.
- Internal face boundaries: fan, radiator, porous jump, wall, and interior.

Cell zones consist of fluids and solids, with porous media treated as a type of fluid zone.

(The internal face boundary conditions are defined on cell faces, which means that they do not have a finite thickness and they provide a means of introducing a step change in flow properties. These boundary condi-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tions are used to implement physical models representing fans, thin porous membranes, and radiators. The "interior" type of internal face zone does not require any input from you.)

In this chapter, the cell zones and boundary conditions listed above will be described in detail, and an explanation of how to set them and where they are most appropriately used will be provided. Note that while periodic boundaries are described in *Periodic Boundary Conditions* (p. 321), additional information about modeling fully-developed periodic flows is provided in *Periodic Flows* (p. 493).

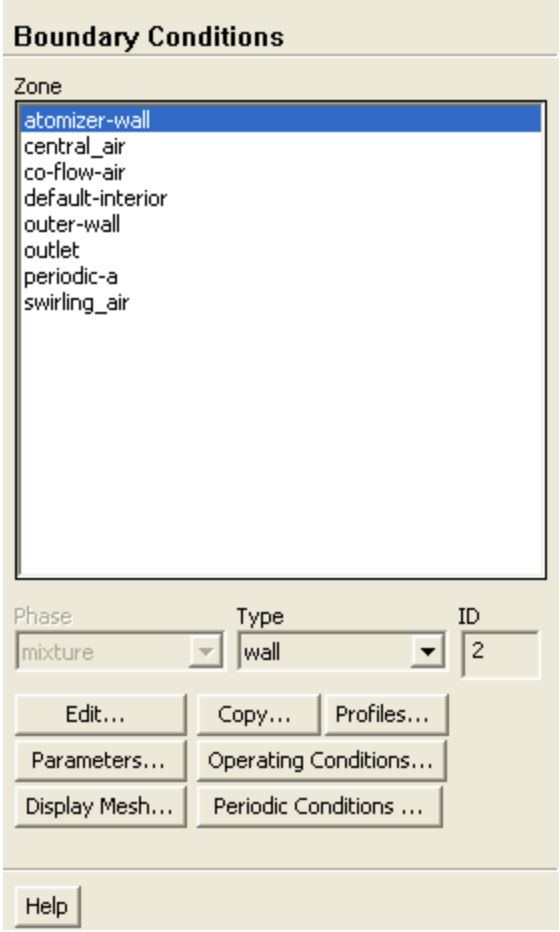
7.1.2. The Cell Zone and Boundary Conditions Task Page

The **Cell Zone Conditions** and **Boundary Conditions** task page (*Figure 7.1* (p. 206)) allows you to change the cell zone or boundary zone type for a given zone and open other dialog boxes to set the cell zone and boundary condition parameters for each zone.

Cell Zone Conditions

Boundary Conditions

Figure 7.1 The Boundary Conditions Task Page



Changing Cell and Boundary Zone Types (p. 207) – *Saving and Reusing Cell Zone and Boundary Conditions* (p. 214) explain how to perform these operations with the **Cell Zone Conditions** or **Boundary Conditions** task page, and how to use the mouse and the graphics display in conjunction with the dialog box.

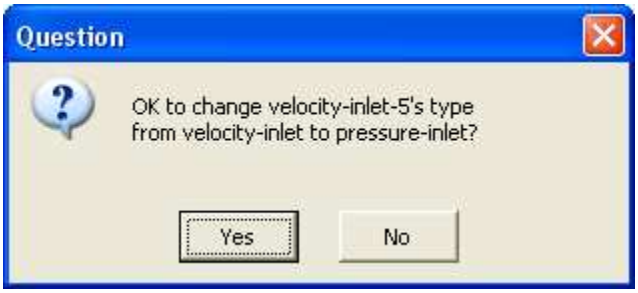
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.3. Changing Cell and Boundary Zone Types

Before you set any cell zone or boundary conditions, you should check the zone types of all boundary zones and change any if necessary. For example, if your mesh includes a pressure inlet, but you want to use a velocity inlet instead, you will need to change the pressure-inlet zone to a velocity-inlet zone.

The steps for changing a zone type are as follows:

1. In the **Cell Zone Conditions** or **Boundary Conditions** task page, select the zone to be changed in the **Zone** list.
2. Choose the correct zone type from the **Type** drop-down list.
3. Confirm the change when prompted by the *Question Dialog Box* (p. 32).



Once you have confirmed the change, the zone type will be changed, the name will change automatically (if the original name was the default name for that zone—see *Changing Cell or Boundary Zone Names* (p. 209)), and the dialog box for setting conditions for the zone will open automatically.

Important

Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. *Creating Conformal Periodic Zones* (p. 191) explains how to create and uncouple periodic zones.

Important

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for changing types is slightly different. See *Steps for Setting Boundary Conditions* (p. 1115) for details.

7.1.3.1. Categories of Zone Types

You should be aware that you can only change zone types within each category listed in *Table 7.1: Zone Types Listed by Category* (p. 207). (Note that a double-sided face zone is a zone that separates two different cell zones or regions.)

Table 7.1 Zone Types Listed by Category

Category	Zone Types
Faces	axis, outflow, mass flow inlet, pressure far-field, pressure inlet, pressure outlet, symmetry, velocity inlet, wall, inlet vent, intake fan, outlet vent, exhaust fan
Double-Sided Faces	fan, interior, porous jump, radiator, wall

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Zone Types
Periodic	periodic
Cells	fluid, solid (porous is a type of fluid cell)

7.1.4. Setting Cell Zone and Boundary Conditions

In ANSYS FLUENT, boundary conditions are associated with zones, not with individual faces or cells. If you want to combine two or more zones that will have the same boundary conditions, see *Merging Zones* (p. 184) for information about merging zones.

To set cell zone and boundary conditions for a particular zone, perform one of the following sequences:

1. Select the zone from the **Zone**list in the **Cell Zone Conditions** or **Boundary Conditions** task page.
2. Click the **Edit...** button.

or

1. Choose the zone in the **Zone** list.
2. Click on the selected zone type from the **Type** drop-down list.

or

1. Double-click on the zone in the **Zone** list.

The dialog box for the selected cell or boundary zone will open, and you can specify the appropriate conditions.

Important

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for setting conditions is slightly different from that described above. See *Steps for Setting Boundary Conditions* (p. 1115) for details.

7.1.5. Copying Cell Zone and Boundary Conditions

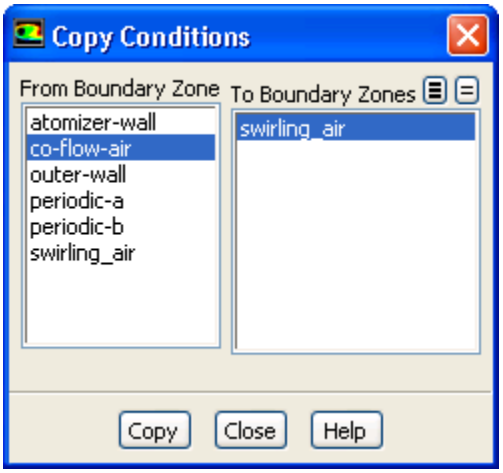
You can copy cell zones and boundary conditions from one zone to other zones of the same type. If, for example, you have several wall zones in your model and they all have the same boundary conditions, you can set the conditions for one wall, and then simply copy them to the others.

The procedure for copying cell zone or boundary conditions is as follows:

1. In the **Cell Zone Conditions** or **Boundary Conditions** task page, click the **Copy...** button. This will open the **Copy Conditions** dialog box (*Figure 7.2* (p. 209)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.2 The Copy Conditions Dialog Box



2. In the **From Cell Zone** or **From Boundary Zone** list, select the zone that has the conditions you want to copy.
3. In the **To Cell Zones** or **To Boundary Zones** list, select the zone or zones to which you want to copy the conditions.
4. Click **Copy**. ANSYS FLUENT will set *all* of the cell zones or boundary conditions for the zones selected in the **To Cell Zones** or **To Boundary Zones** list to be the same as the conditions for the zone selected in the **From Cell Zone** or **From Boundary Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Note that you cannot copy conditions from external walls to internal (i.e., two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

Important

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for copying boundary conditions is slightly different. See *Steps for Copying Cell Zone and Boundary Conditions* (p. 1119) for details.

7.1.6. Changing Cell or Boundary Zone Names

The default name for a zone is its type plus an ID number (e.g., **pressure-inlet-7**). In some cases, you may want to assign more descriptive names to the boundary zones. If you have two pressure-inlet zones, for example, you might want to rename them **small-inlet** and **large-inlet**. (Changing the name of a zone will *not* change its type. Instructions for changing a zone's type are provided in *Changing Cell and Boundary Zone Types* (p. 207).)

To rename a zone, follow these steps:

1. Select the zone to be renamed in the **Zones** list in the **Cell Zone Conditions** or **Boundary Conditions** task page.
2. Click **Edit...** to open the dialog box for the selected zone.
3. Enter a new name under **Zone Name**.
4. Click the **OK** button.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that if you specify a new name for a zone and then change its type, the name you specified will be retained; the automatic name change that accompanies a change in type occurs only if the name of the zone is its type plus its ID.

7.1.7. Defining Non-Uniform Cell Zones and Boundary Conditions

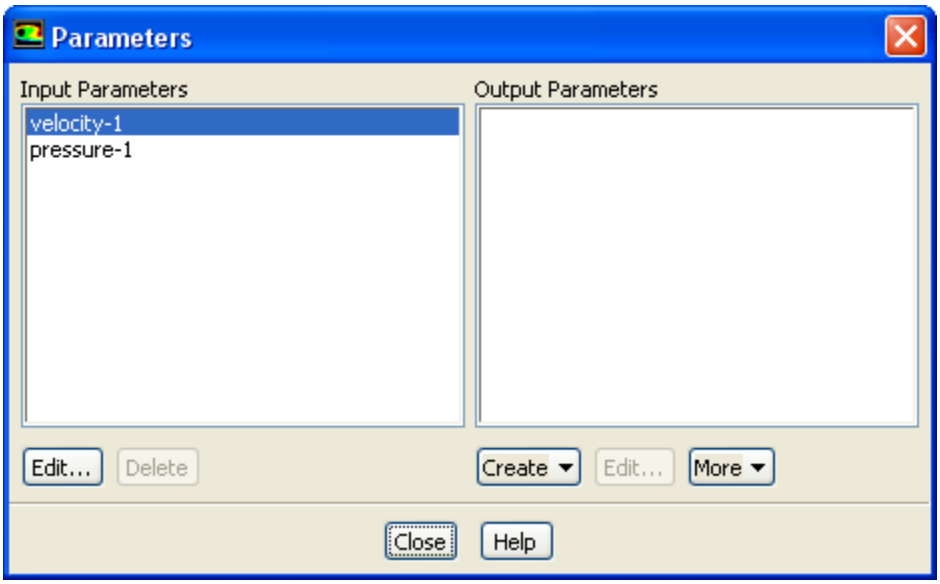
Most conditions at each type of boundary zone can be defined as profile functions instead of constant values. You can use a profile contained in an externally generated profile file, or a function that you create using a user-defined function (UDF). Profiles are described in *Profiles* (p. 366), and user-defined functions are described in the separate UDF Manual.

7.1.8. Defining and Viewing Parameters

You can define a series of cases based on a set of parametric values. These parameters may be defined for numeric cell zone and boundary condition settings. This is especially useful if you are using Workbench and performing parametric studies (optimization), comparing cases with different boundary settings. Information about this feature can be found in the Getting Started with ANSYS Workbench manual, in the Parameter Manager chapter. If you are not running ANSYS FLUENT through Workbench, then you can use the parameter settings to define the same boundary condition value to different boundaries having the same units.

The parameters that you have defined in the various boundary condition dialog boxes are accessed by clicking the **Parameters...** button in the **Cell Zone Conditions** or **Boundary Conditions** task page. The **Parameters** dialog box will open, as shown in *Figure 7.3* (p. 210), listing all of the input parameters that you have created in the various boundary condition dialog boxes.

Figure 7.3 The Parameters Dialog Box



In the **Parameters** dialog box, you can

- Edit the input properties using the **Input Parameter Properties** dialog box, which is the same dialog box used to create parameters. This dialog box can also be accessed by selecting **New Input Parameter...** from the drop-down lists in the boundary conditions dialog boxes, as described later in this section.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you are using ANSYS FLUENT in ANSYS Workbench, you cannot edit the input parameters, you can only view them. For more information, see the separate ANSYS FLUENT in **Workbench** User's Guide.

- Delete input parameters which are not assigned to a setting.
- Create output parameters. These are single values generated by existing reports. The types of output that may be generated are **Fluxes...**, **Forces...**, **Surface Integrals...**, and **Volume Integrals...**. These output parameters are discussed in greater detail in *Creating Output Parameters* (p. 1509).
- More options exist under the **More** menu:

  Delete
  displays a message in a dialog box, prompting you for a response to confirm the deletion of the output parameter.

  Rename
  allows you to edit the name of the output parameter through the **Rename** dialog box.

  Print to Console
  will output values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

  Print All to Console
  outputs the values from all output parameters to the console window.

  Write...
  allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

  Write All...
  prompts you for a file name and then writes the values for all of the output parameters to a file.

Important

Changing the units for a quantity changes the value for any input parameter using that quantity.

Note

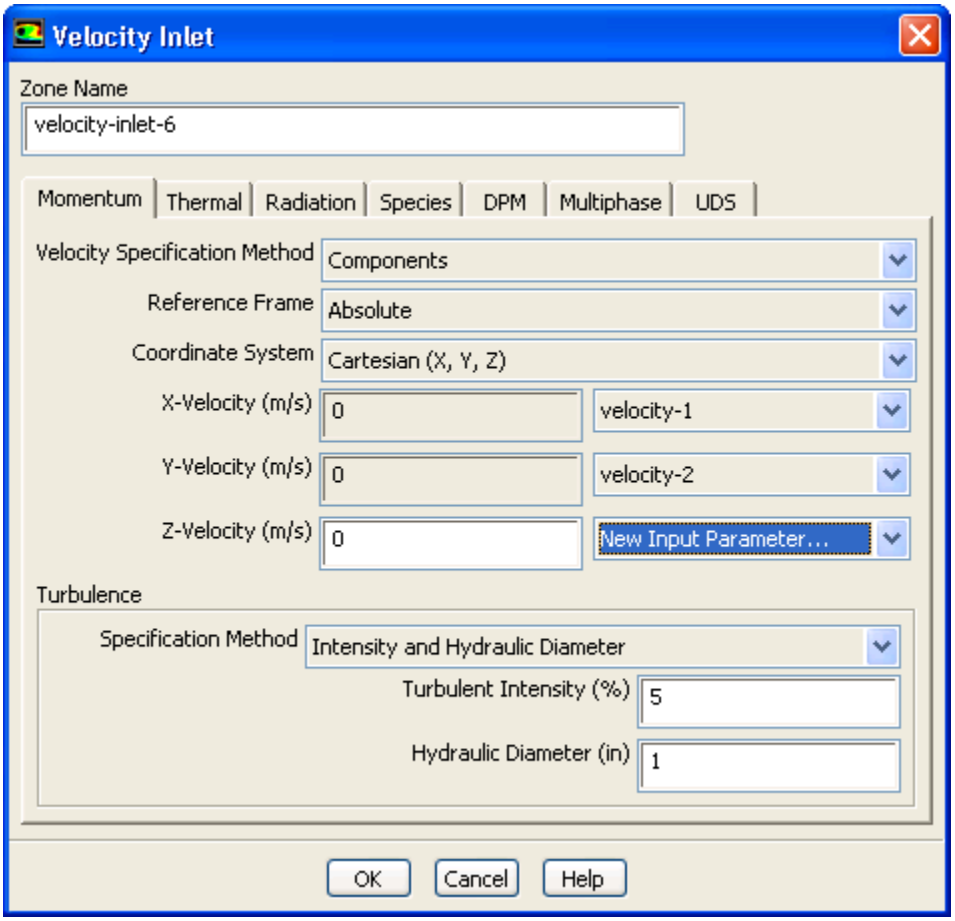
Source term parameters can only be specified using SI units. The default parameter name will not include the unit quantity name. Even if you have selected units other than SI for the source term parameters, you must still provide values assuming SI units for the (source-term) parameters in *both* FLUENT and Workbench.

7.1.8.1. Creating a New Parameter

You can create a new cell zone or boundary condition parameter, as shown in *Figure 7.4* (p. 212).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

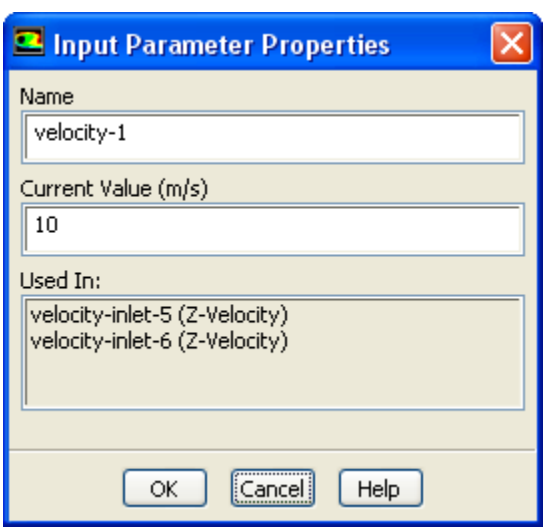
Figure 7.4 The New Input Parameter... Selection



When you select **New Input Parameter...** from the drop-down list, the **Input Parameter Properties** dialog box (*Figure 7.5* (p. 212)) will open where you will

- Enter the **Name** of the parameter.
- Specify the **Current Value** as a constant.

Figure 7.5 The Input Parameter Properties Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the parameters are defined in the **Input Parameter Properties** dialog box, the name of the parameter will appear in the drop-down list of the property you are defining, as seen in *Figure 7.4* (p. 212).

7.1.9. Selecting Cell or Boundary Zones in the Graphics Display

Whenever you need to select a zone in the **Cell Zone Conditions** or **Boundary Conditions** task page, you can use the mouse in the graphics window to choose the appropriate zone. This feature is particularly useful if you are setting up a problem for the first time, or if you have two or more zones of the same type and you want to determine the zone IDs (i.e., figure out which zone is which). To use this feature, do the following:

1. Display the mesh using the *Mesh Display Dialog Box* (p. 1635).
2. Use the mouse probe button (the right button, by default—see *Controlling the Mouse Button Functions* (p. 1427) to modify the mouse button functions) to click on a cell or boundary zone in the graphics window.

The zone you select in the graphics display will automatically be selected in the **Zone** list in the **Cell Zone Conditions** or **Boundary Conditions** task page, and its name and ID will be printed in the console window.

However, if you prefer to select the surfaces to display using the **Mesh Display** dialog box, then simply click the **Display Mesh...** button in the **Cell Zone Conditions** or **Boundary Conditions** task page.

Cell Zone Conditions → **Display Mesh...**

Boundary Conditions → **Display Mesh...**

Detailed information about the **Mesh Display** dialog box can be found in *Displaying the Mesh* (p. 1379).

7.1.10. Operating and Periodic Conditions

The **Cell Zone Conditions** and **Boundary Conditions** task pages allow you to access the **Operating Conditions** dialog box, where you can set the operating pressure, reference pressure location, include the effects of gravity, and specify other operating variables, as discussed in *Modeling Basic Fluid Flow* (p. 485).

Cell Zone Conditions → **Operating Conditions...**

Boundary Conditions → **Operating Conditions...**

The **Periodic Conditions** dialog box can be accessed from the **Boundary Conditions** task page. For a detailed description of this dialog box's inputs, refer to *Periodic Flows* (p. 493).

Boundary Conditions → **Periodic Conditions...**

7.1.11. Highlighting Selected Boundary Zones

To highlight a selected boundary zone of a model which is displayed in the graphics window, enable the **Highlight Zone** option in the **Boundary Conditions** task page.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that this is only applicable to 3D cases and is available only in the **Boundary Conditions** task page.

There are two ways in which you can highlight a boundary zone in the graphics window, after enabling the **Highlight Zone** option:

- Select the zone by highlighting it in the **Zone** list, in the **Boundary Conditions** task page.
- Use the mouse-probe button and select the zone in the graphics window.

The selected zone in the graphics window will be highlighted in a cyan color, which is the default color. You can change the color using the following text command:

`display` → `set` → `colors` → `highlight-color`

If you want to highlight a boundary zone which is not displayed in the graphics window, ANSYS FLUENT will display that boundary zone in the graphics window and then highlight it. If you disable the **Highlight Zone** option, the displayed boundary zone will be removed from the scene (graphics window) and your model will be redrawn to its original view.

7.1.12. Saving and Reusing Cell Zone and Boundary Conditions

You can save cell zone and boundary conditions to a file so that you can use them to specify the same conditions for a different case, as described in *Reading and Writing Boundary Conditions* (p. 72).

7.2. Cell Zone Conditions

Cell zones consist of fluids and solids. Porous zones in ANSYS FLUENT are treated as fluid zones. A detailed description of the various cell zones is given in the sections that follow.

7.2.1. Fluid Conditions
7.2.2. Solid Conditions
7.2.3. Porous Media Conditions
7.2.4. Fixing the Values of Variables
7.2.5. Defining Mass, Momentum, Energy, and Other Sources

7.2.1. Fluid Conditions

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid material. You must indicate which material the fluid zone contains so that the appropriate material properties will be used.

Important

If you are modeling multiphase flow, you will not specify the materials here; you will choose the phase material when you define the phases, as described in *Defining the Phases for the VOF Model* (p. 1136).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you are modeling species transport and/or combustion, you can specify the material as either a mixture or a fluid. The mixture material has to be the same as that specified in the **Species Model** dialog box when you enable the model. The fluid zones, being of different material types, must not be contiguous.

Optional inputs allow you to set sources or fixed values of mass, momentum, heat (temperature), turbulence, species, and other scalar quantities. You can also define motion for the fluid zone. If there are rotationally periodic boundaries adjacent to the fluid zone, you will need to specify the rotation axis. If you are modeling turbulence using one of the k- ε models, the k- ω model, or the Spalart-Allmaras model, you can choose to define the fluid zone as a laminar flow region. If you are modeling radiation using the DO model, you can specify whether or not the fluid participates in radiation.

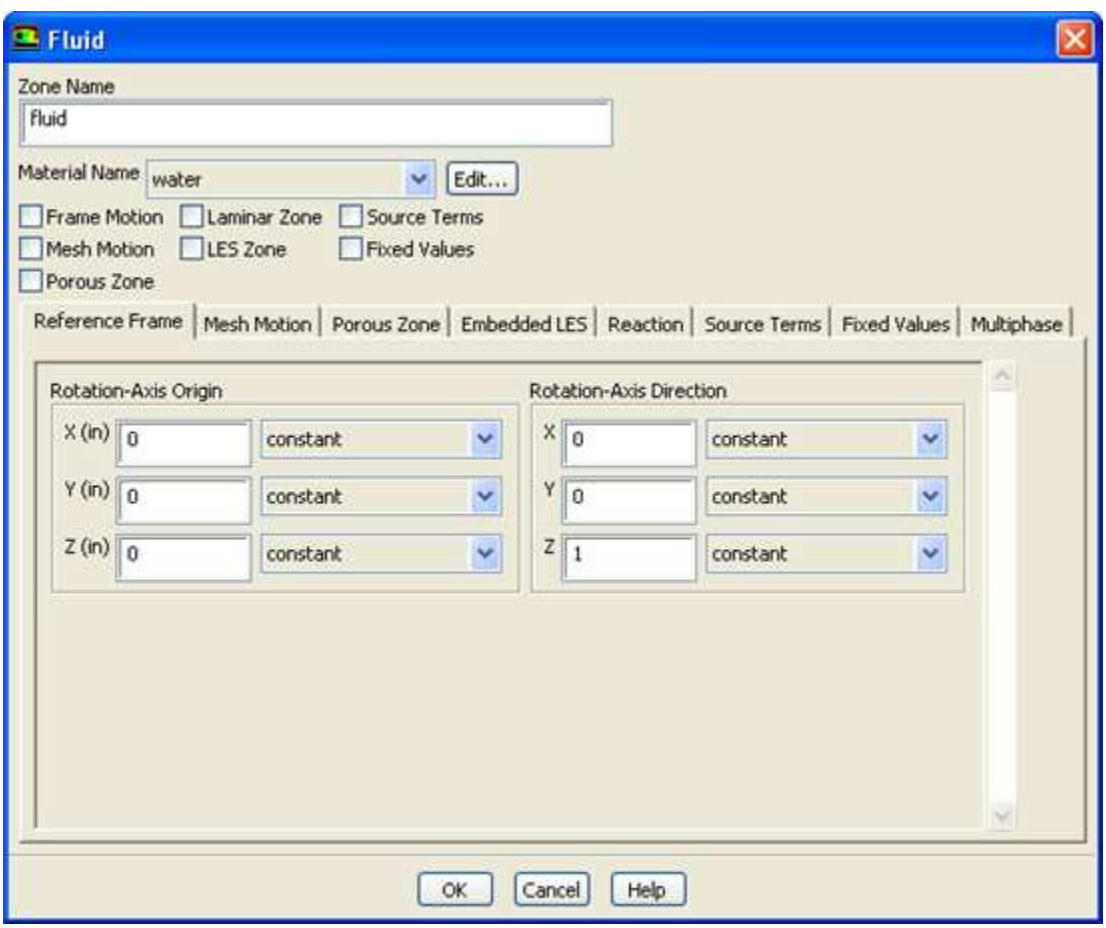
Important

For information about porous zones, see *Porous Media Conditions* (p. 221).

7.2.1.1. Inputs for Fluid Zones

You will set all fluid conditions in the *Fluid Dialog Box* (p. 1796) (*Figure 7.6* (p. 215)), which is accessed from the **Cell Zone Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Figure 7.6 The Fluid Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.1.1.1. Defining the Fluid Material

To define the material contained in the fluid zone, select the appropriate item in the **Material Name** drop-down list. This list will contain all fluid materials database) in the *Create/Edit Materials Dialog Box* (p. 1736). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

Important

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid zones will be the material you specified in the *Species Model Dialog Box* (p. 1681). For multiphase flows, the materials are specified when you define the phases, as described in *Defining the Phases for the VOF Model* (p. 1136).

7.2.1.1.2. Defining Sources

If you wish to define a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone, you can do so by enabling the **Source Terms** option. See *Defining Mass, Momentum, Energy, and Other Sources* (p. 244) for details.

7.2.1.1.3. Defining Fixed Values

If you wish to fix the value of one or more variables in the fluid zone, rather than computing them during the calculation, you cando so by enabling the **Fixed Values** option. See *Fixing the Values of Variables* (p. 241) for details.

7.2.1.1.4. Specifying a Laminar Zone

When you are calculating a turbulent flow, it is possible to "turn off" turbulence modeling in a specific fluid zone. To disable turbulence modeling, turn on the **Laminar Zone** option in the **Fluid** dialog box. This will disable the turbulence production, but transport the turbulence quantities. If, in addition, you want to set the turbulent viscosity to zero you can do that using the text command `define/boundary-conditions/fluid`. You will be asked if you want to `Set Turbulent Viscosity to zero within laminar zone?`. If your response is `yes`, ANSYS FLUENT will set both the production term in the turbulence transport equation and μ_t to zero. This is useful if you know that the flow in a certain region is laminar. For example, if you know the location of the transition point on an airfoil, you can create a laminar/turbulent transition boundary where the laminar cell zone borders the turbulent cell zone. This feature allows you to model turbulent transition on the airfoil.

Disabling turbulence modeling in a fluid zone, can be applied to all the turbulence models except the Large Eddy Simulation (LES) model.

7.2.1.1.5. Specifying a Reaction Mechanism

If you are modeling species transport with reactions, you can enable a reaction mechanism in a fluid zone by turning on the **Reaction** option and selecting an available mechanism from the **Reaction Mechanism** drop-down list. See *Defining Zone-Based Reaction Mechanisms* (p. 816) for more information about defining reaction mechanisms.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.1.1.6. Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the fluid zone or if the zone is rotating, either the mesh or its reference frame, you must specify the rotation axis. To define the axis for a moving reference frame problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Reference Frame** tab. To define the axis for a moving mesh problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Mesh Motion** tab.

Note

If a frame motion and a mesh motion are specified at the same zone and this zone has rotationally periodic boundaries adjacent to it, then both axes have to be coaxial. Otherwise, the periodicity assumption is not valid and you will receive a warning message. In addition, the mesh check will fail.

The cell zone axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction** inputs for the frame of reference and the mesh motion. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

7.2.1.1.7. Defining Zone Motion

To define zone motion for a rotating or translating reference frame, enable the **Frame Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Reference Frame** tab.

To define zone motion for a sliding mesh, enable the **Mesh Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Mesh Motion** tab. See *Setting Up the Sliding Mesh Problem* (p. 541) for details.

For cases where you have a moving zone within another moving zone, select the cell zone carrying the primary motion from the **Relative To Cell Zone** drop-down list under the **Reference Frame** or the **Mesh Motion** tab. Note that the list will consist of all moving cell zones with an absolute motion specification, excluding the current cell zone. If you select **absolute**, then the velocity and rotation components are specified in an absolute reference frame. This is the default value. If no moving zones are present in the simulation, then **absolute** will be the only available selection. See The Multiple Reference Frame Model for more information.

For problems that include linear, translational motion of the fluid zone, specify the **Translational Velocity** by setting the **X**, **Y**, and **Z** components under the **Mesh Motion** tab. For problems that include rotational motion, specify the rotational **Speed** under **Rotational Velocity**. The rotation axis is defined as described above. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in *Defining Transient Cell Zone and Boundary Conditions* (p. 376), or a UDF macro (DEFINE_TRANSIENT_PROFILE). Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the frame motion individually. However, you can also specify the frame motion using a user-defined function. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you need to switch between the MRF and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

See *Modeling Flows with Moving Reference Frames* (p. 513) for details about modeling flows in moving reference frames. Details about the frame motion UDF can be found in DEFINE_ZONE_MOTION in the UDF Manual.

7.2.1.1.8. Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the fluid zone participates in radiation using the **Participates in Radiation** option. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

7.2.2. Solid Conditions

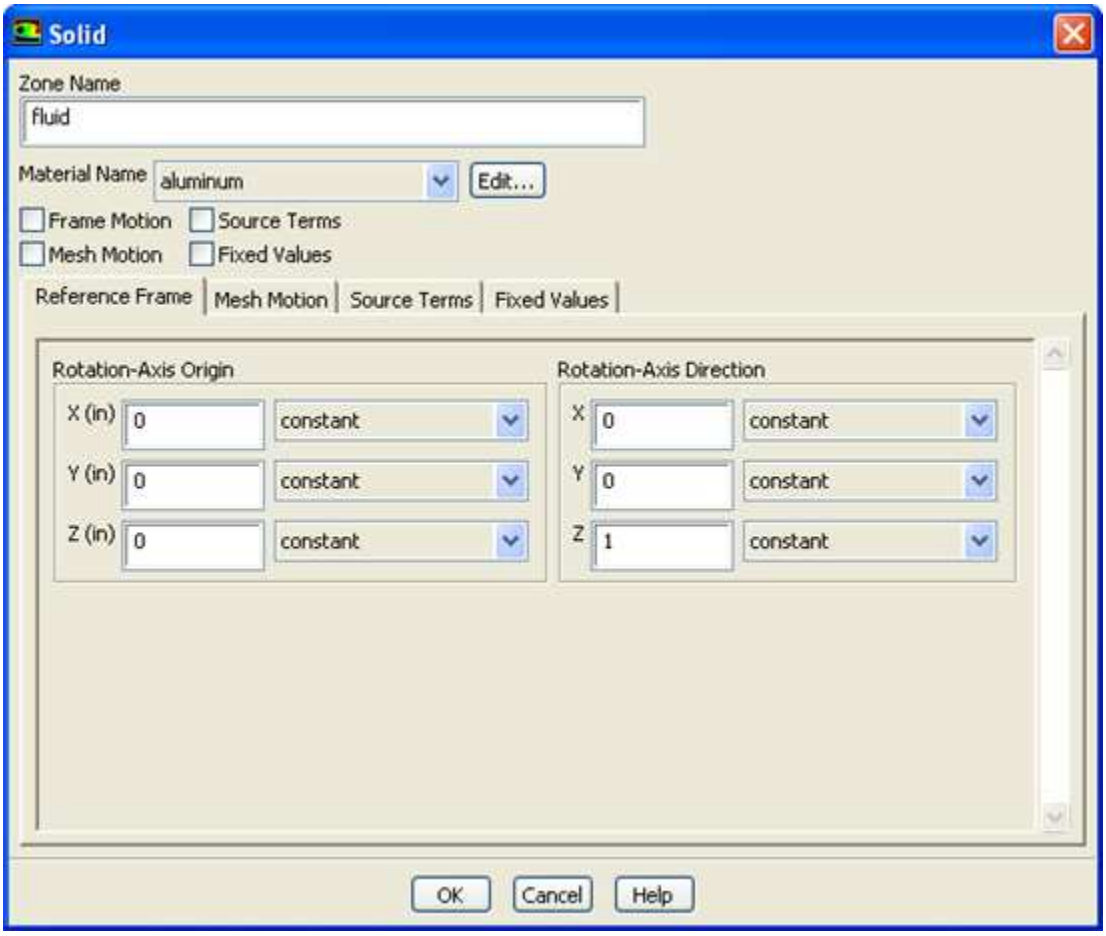
A "solid" zone is a group of cells for which only a heat conduction problem is solved; no flow equations are solved. The material being treated as a solid may actually be a fluid, but it is assumed that no convection is taking place. The only required input for a solid zone is the type of solid material. You must indicate which material the solid zone contains so that the appropriate material properties will be used. Optional inputs allow you to set a volumetric heat generation rate (heat source) or a fixed value of temperature. You can also define motion for the solid zone. If there are rotationally periodic boundaries adjacent to the solid zone, you will need to specify the rotation axis. If you are modeling radiation using the DO model, you can specify whether or not the solid material participates in radiation.

7.2.2.1. Inputs for Solid Zones

You will set all solid conditions in the *Solid Dialog Box* (p. 1802) (*Figure 7.7* (p. 219)), which is opened from the **Cell Zone Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.7 The Solid Dialog Box



7.2.2.1.1. Defining the Solid Material

To define the material contained in the solid zone, select the appropriate item in the **Material Name** drop-down list. This list will contain all solid materials database) in the *Create/Edit Materials Dialog Box* (p. 1736). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

7.2.2.1.2. Defining a Heat Source

If you wish to define a source of heat within the solid zone, you can do so by enabling the **Source Terms** option. See *Defining Mass, Momentum, Energy, and Other Sources* (p. 244) for details.

7.2.2.1.3. Defining a Fixed Temperature

If you wish to fix the value of temperature in the solid zone, rather than computing it during the calculation, you can do so by enabling the **Fixed Values** option. See *Fixing the Values of Variables* (p. 241) for details.

7.2.2.1.4. Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the fluid zone or if the zone is rotating, either the mesh or its reference frame, you must specify the rotation axis. To define the axis for a moving reference frame problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Reference Frame** tab.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define the axis for a moving mesh problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Mesh Motion** tab.

Note

If a frame motion and a mesh motion are specified at the same zone and this zone has rotationally periodic boundaries adjacent to it, then both axes have to be coaxial. Otherwise, the periodicity assumption is not valid and you will receive a warning message. In addition, the mesh check will fail.

The cell zone axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction** inputs for the frame of reference and the mesh motion. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

7.2.2.1.5. Defining Zone Motion

To define zone motion for a rotating or translating reference frame, enable the **Frame Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Reference Frame** tab.

To define zone motion for a sliding mesh, enable the **Mesh Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Mesh Motion** tab. See *Setting Up the Sliding Mesh Problem* (p. 541) for details.

For cases where you have a moving zone within another moving zone, select the cell zone carrying the primary motion from the **Relative To Cell Zone** drop-down list under the **Reference Frame** or the **Mesh Motion** tab. Note that the list will consist of all moving cell zones with an absolute motion specification, excluding the current cell zone. If you select **absolute**, then the velocity and rotation components are specified in an absolute reference frame. This is the default value. If no moving zones are present in the simulation, then **absolute** will be the only available selection. See The Multiple Reference Frame Model for more information.

For problems that include linear, translational motion of the fluid zone, specify the **Translational Velocity** by setting the **X**, **Y**, and **Z** components under the **Mesh Motion** tab. For problems that include rotational motion, specify the rotational **Speed** under **Rotational Velocity**. The rotation axis is defined as described above. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in *Defining Transient Cell Zone and Boundary Conditions* (p. 376), or a UDF macro (DEFINE_TRANSIENT_PROFILE). Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the frame motion individually. However, you can also specify the frame motion using a user-defined function. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

See *Modeling Flows with Moving Reference Frames* (p. 513) for details about modeling flows in moving reference frames. Details about the frame motion UDF can be found in DEFINE_ZONE_MOTION in the UDF Manual.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you need to switch between the MRF and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

7.2.2.1.6. Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the solid material participates in radiation using the **Participates in Radiation** option. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

7.2.3. Porous Media Conditions

The porous media model can be used for a wide variety of single phase and multiphase problems, including flow through packed beds, filter papers, perforated plates, flow distributors, and tube banks. When you use this model, you define a cell zone in which the porous media model is applied and the pressure loss in the flow is determined via your inputs as described in *Momentum Equations for Porous Media* (p. 222). Heat transfer through the medium can also be represented, subject to the assumption of thermal equilibrium between the medium and the fluid flow, as described in *Treatment of the Energy Equation in Porous Media* (p. 224).

A 1D simplification of the porous media model, termed the "porous jump," can be used to model a thin membrane with known velocity/pressure-drop characteristics. The porous jump model is applied to a face zone, not to a cell zone, and should be used (instead of the full porous media model) whenever possible because it is more robust and yields better convergence. See *Porous Jump Boundary Conditions* (p. 337) for details.

7.2.3.1. Limitations and Assumptions of the Porous Media Model

The porous media model incorporates an empirically determined flow resistance in a region of your model defined as "porous". In essence, the porous media model is nothing more than an added momentum sink in the governing momentum equations. As such, the following modeling assumptions and limitations should be readily recognized:

- Since the volume blockage that is physically present is not represented in the model, by default ANSYS FLUENT uses and reports a superficial velocity inside the porous medium, based on the volumetric flow rate, to ensure continuity of the velocity vectors across the porous medium interface. This superficial velocity formulation does not take porosity into account when calculating the convection and diffusion terms of the transport equations. You can choose to use a more accurate alternative, in which the true (physical) velocity is calculated inside the porous medium and porosity is included in the differential terms of the transport equations. See *Modeling Porous Media Based on Physical Velocity* (p. 237) for details. In a multiphase flow system, all phases share the same porosity.
- The effect of the porous medium on the turbulence field is only approximated. See *Treatment of Turbulence in Porous Media* (p. 225) for details.
- In general, the ANSYS FLUENT porous medium model, for both single phase and multiphase, assumes the porosity is isotropic, and it can vary with space and time.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Superficial Velocity Formulation and the Physical Velocity Formulation are available for multiphase porous media. See *User Inputs for Porous Media* (p. 225) for details.
- The porous media momentum resistance and heat source terms are calculated separately on each phase. See *Momentum Equations for Porous Media* (p. 222) for details.
- The interactions between a porous medium and shock waves are not considered.
- By default, ANSYS FLUENT assumes thermal equilibrium between the porous media solids and multiphase fluid flows. The solids temperature is thus estimated by phase temperatures. However, the solids temperature can also be calculated by a UDS equation (*User-Defined Scalar (UDS) Transport Equations* (p. 485)).
- When applying the porous media model in a moving reference frame, ANSYS FLUENT will either apply the relative reference frame or the absolute reference frame when you enable the **Relative Velocity Resistance Formulation**. This allows for the correct prediction of the source terms.

7.2.3.2. Momentum Equations for Porous Media

The porous media models for single phase flows and multiphase flows use the **Superficial Velocity Porous Formulation** as the default. ANSYS FLUENT calculates the superficial phase or mixture velocities based on the volumetric flow rate in a porous region. The porous media model is described in the following sections for single phase flow, however, it is important to note the following for multiphase flow:

- In the Eulerian multiphase model (Eulerian Model Theory in the Theory Guide), the general porous media modeling approach, physical laws, and equations described below are applied to the corresponding phase for mass continuity, momentum, energy, and all the other scalar equations.
- The **Superficial Velocity Porous Formulation** generally gives good representations of the bulk pressure loss through a porous region. However, since the superficial velocity values within a porous region remain the same as those outside the porous region, it cannot predict the velocity increase in porous zones and thus limits the accuracy of the model.

Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term (Darcy, the first term on the right-hand side of *Equation 7–1* (p. 222), and an inertial loss term (the second term on the right-hand side of *Equation 7–1* (p. 222))

$$S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho |v| v_j \right) \qquad \textbf{(7–1)}$$

where S_i is the source term for the ith (x, y, or z) momentum equation, $|v|$ is the magnitude of the velocity and D and C are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell.

To recover the case of simple homogeneous porous media

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right) \qquad \textbf{(7–2)}$$

where α is the permeability and C_2 is the inertial resistance factor, simply specify D and C as diagonal matrices with $1/\alpha$ and C_2, respectively, on the diagonals (and zero for the other elements).

ANSYS FLUENT also allows the source term to be modeled as a power law of the velocity magnitude:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$S_i = -C_0 |v|^{C_1} = -C_0 |v|^{(C_1 - 1)} v_i \quad \textbf{(7–3)}$$

where C_0 and C_1 are user-defined empirical coefficients.

Important

In the power-law model, the pressure drop is isotropic and the units for C_0 are SI.

7.2.3.2.1. Darcy's Law in Porous Media

In laminar flows through porous media, the pressure drop is typically proportional to velocity and the constant C_2 can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy's Law:

$$\nabla p = -\frac{\mu}{\alpha} \vec{v} \quad \textbf{(7–4)}$$

The pressure drop that ANSYS FLUENT computes in each of the three (x, y, z) coordinate directions within the porous region is then

$$\Delta p_x = \sum_{j=1}^{3} \frac{\mu}{\alpha_{xj}} v_j \Delta n_x$$

$$\Delta p_y = \sum_{j=1}^{3} \frac{\mu}{\alpha_{yj}} v_j \Delta n_y \quad \textbf{(7–5)}$$

$$\Delta p_z = \sum_{j=1}^{3} \frac{\mu}{\alpha_{zj}} v_j \Delta n_z$$

where $1/\alpha_{ij}$ are the entries in the matrix D in *Equation 7–1* (p. 222), v_j are the velocity components in the x, y, and z directions, and Δn_x, Δn_y, and Δn_z are the thicknesses of the medium in the x, y, and z directions.

Here, the thickness of the medium (Δn_x, Δn_y, or Δn_z) is the *actual* thickness of the porous region in your model. Thus if the thicknesses used in your model differ from the actual thicknesses, you must make the adjustments in your inputs for $1/\alpha_{ij}$.

7.2.3.2.2. Inertial Losses in Porous Media

At high flow velocities, the constant C_2 in *Equation 7–1* (p. 222) provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

If you are modeling a perforated plate or tube bank, you can sometimes eliminate the permeability term and use the inertial loss term alone, yielding the following simplified form of the porous media equation:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\nabla p = -\sum_{j=1}^{3} C_{2_{ij}} \left(\frac{1}{2} \rho v_j |v| \right) \tag{7–6}$$

or when written in terms of the pressure drop in the x, y, z directions:

$$\begin{aligned} \Delta p_x &\approx \sum_{j=1}^{3} C_{2_{xj}} \Delta n_x \frac{1}{2} \rho v_j |v| \\ \Delta p_y &\approx \sum_{j=1}^{3} C_{2_{yj}} \Delta n_y \frac{1}{2} \rho v_j |v| \\ \Delta p_z &\approx \sum_{j=1}^{3} C_{2_{zj}} \Delta n_z \frac{1}{2} \rho v_j |v| \end{aligned} \tag{7–7}$$

Again, the thickness of the medium (Δn_x, Δn_y, or Δn_z) is the thickness you have defined in your model.

7.2.3.3. Treatment of the Energy Equation in Porous Media

ANSYS FLUENT solves the standard energy transport equation (Equation 5–1 in the Theory Guide) in porous media regions with modifications to the conduction flux and the transient terms only. In the porous medium, the conduction flux uses an effective conductivity and the transient term includes the thermal inertia of the solid region on the medium:

$$\frac{\partial}{\partial t}\left(\gamma \rho_f E_f + (1-\gamma)\rho_s E_s\right) + \nabla \cdot \left(\vec{v}\left(\rho_f E_f + p\right)\right) = \nabla \cdot \left[k_{eff} \nabla T - (\sum_i h_i J_i) + (\bar{\bar{\tau}} \cdot \vec{v})\right] + S_f^h \tag{7–8}$$

where

E_f = total fluid energy

E_s = total solid medium energy

γ = porosity of the medium

k_{eff} = effective thermal conductivity of the medium

S_f^h = fluid enthalpy source term

7.2.3.3.1. Effective Conductivity in the Porous Medium

The effective thermal conductivity in the porous medium, k_{eff}, is computed by ANSYS FLUENT as the volume average of the fluid conductivity and the solid conductivity:

$$k_{eff} = \gamma k_f + (1-\gamma) k_s \tag{7–9}$$

where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

γ = porosity of the medium

k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_s = solid medium thermal conductivity

The fluid thermal conductivity k_f and the solid thermal conductivity k_s can be computed via user-defined functions.

The anisotropic effective thermal conductivity can also be specified via user-defined functions. In this case, the isotropic contributions from the fluid, γk_f, are added to the diagonal elements of the solid anisotropic thermal conductivity matrix.

7.2.3.4. Treatment of Turbulence in Porous Media

ANSYS FLUENT will, by default, solve the standard conservation equations for turbulence quantities in the porous medium. In this default approach, turbulence in the medium is treated as though the solid medium has no effect on the turbulence generation or dissipation rates. This assumption may be reasonable if the medium's permeability is quite large and the geometric scale of the medium does not interact with the scale of the turbulent eddies. In other instances, however, you may want to suppress the effect of turbulence in the medium.

If you are using one of the turbulence models (with the exception of the Large Eddy Simulation (LES) model), you can suppress the effect of turbulence in a porous region by setting the turbulent contribution to viscosity, μ_t, equal to zero. When you choose this option, ANSYS FLUENT will transport the inlet turbulence quantities through the medium, but their effect on the fluid mixing and momentum will be ignored. In addition, the generation of turbulence will be set to zero in the medium. This modeling strategy is enabled by turning on the **Laminar Zone** option in the *Fluid Dialog Box* (p. 1796). Enabling this option implies that μ_t is zero and that generation of turbulence will be zero in this porous zone. Disabling the option (the default) implies that turbulence will be computed in the porous region just as in the bulk fluid flow. Refer to *Specifying a Laminar Zone* (p. 216) for details about using the **Laminar Zone** option.

7.2.3.5. Effect of Porosity on Transient Scalar Equations

For transient porous media calculations, the effect of porosity on the time-derivative terms is accounted for in all scalar transport equations and the continuity equation. When the effect of porosity is taken into account, the time-derivative term becomes $\frac{\partial}{\partial t}(\gamma\rho\phi)$, where ϕ is the scalar quantity (k, ε, etc.) and γ is the porosity.

The effect of porosity is enabled automatically for transient calculations, and the porosity is set to 1 by default.

7.2.3.6. User Inputs for Porous Media

When you are modeling a porous region, the only additional inputs for the problem setup are as follows. Optional inputs are indicated as such.

1. Define the porous zone.
2. Define the porous velocity formulation in the **Cell Zone Conditions** task page. (optional)
3. Identify the fluid material flowing through the porous medium.
4. Enable reactions for the porous zone, if appropriate, and select the reaction mechanism.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

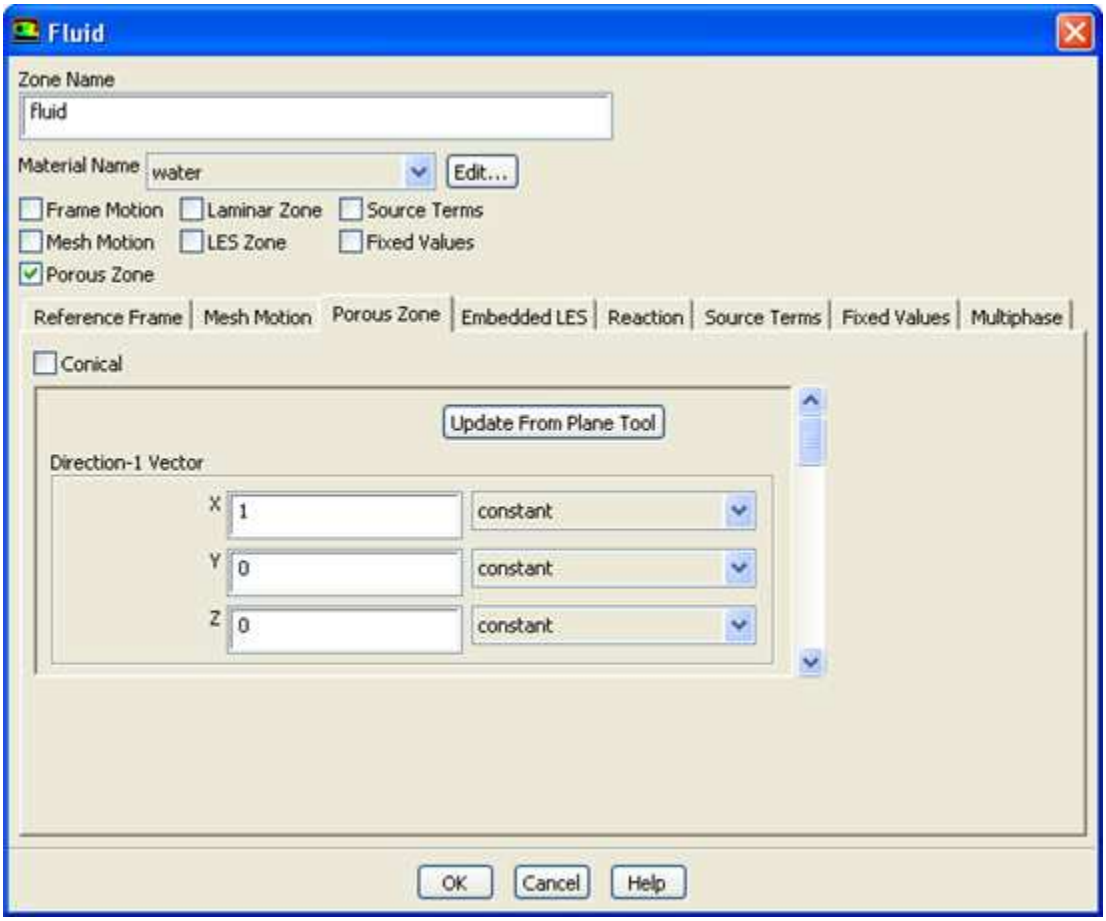
5. Enable the **Relative Velocity Resistance Formulation**. By default, this option is already enabled and takes the moving porous media into consideration (as described in *Including the Relative Velocity Resistance Formulation* (p. 228)).
6. Set the viscous resistance coefficients (D_{ij} in *Equation 7–1* (p. 222), or $1/\alpha$ in *Equation 7–2* (p. 222)) and the inertial resistance coefficients (C_{ij} in *Equation 7–1* (p. 222), or C_2 in *Equation 7–2* (p. 222)), and define the direction vectors for which they apply. Alternatively, specify the coefficients for the power-law model.
7. Specify the porosity of the porous medium.
8. Select the material contained in the porous medium (required only for models that include heat transfer). Note that the specific heat capacity, C_p, for the selected material in the porous zone can only be entered as a constant value.
9. Set the volumetric heat generation rate in the non-solid portion of the porous medium (or any other sources, such as mass or momentum). (optional)
10. Set any fixed values for solution variables in the fluid region (optional).
11. Suppress the turbulent viscosity in the porous region, if appropriate.
12. Specify the rotation axis and/or zone motion, if relevant.

Methods for determining the resistance coefficients and/or permeability are presented below. If you choose to use the power-law approximation of the porous-media momentum source term, you will enter the coefficients C_0 and C_1 in *Equation 7–3* (p. 223) instead of the resistance coefficients and flow direction.

You will set all parameters for the porous medium in the *Fluid Dialog Box* (p. 1796) (*Figure 7.8* (p. 227)), which is opened from the **Cell Zone Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.8 The Fluid Dialog Box for a Porous Zone



7.2.3.6.1. Defining the Porous Zone

As mentioned in *Overview* (p. 205), a porous zone is modeled as a special type of fluid zone. To indicate that the fluid zone is a porous region, enable the **Porous Zone** option in the **Fluid** dialog box. The dialog box will expand to show the porous media inputs (as shown in *Figure 7.8* (p. 227)).

7.2.3.6.2. Defining the Porous Velocity Formulation

The **Cell Zone Conditions** task page contains a **Porous Formulation** region where you can instruct ANSYS FLUENT to use either a superficial or physical velocity in the porous medium simulation. By default, the velocity is set to **Superficial Velocity**. For details about using the **Physical Velocity** formulation, see *Modeling Porous Media Based on Physical Velocity* (p. 237).

7.2.3.6.3. Defining the Fluid Passing Through the Porous Medium

To define the fluid that passes through the porous medium, select the appropriate fluid in the **Material Name** drop-down list in the *Fluid Dialog Box* (p. 1796). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid/porous zones will be the material you specified in the *Species Model Dialog Box* (p. 1681). For multiphase flows, the materials are specified when you define the phases, as described in *Defining the Phases for the VOF Model* (p. 1136).

7.2.3.6.4. Enabling Reactions in a Porous Zone

If you are modeling species transport with reactions, you can enable reactions in a porous zone by turning on the **Reaction** option in the **Fluid** dialog box and selecting a mechanism in the **Reaction Mechanism** drop-down list.

If your mechanism contains wall surface reactions, you will also need to specify a value for the **Surface-to-Volume Ratio**. This value is the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, ANSYS FLUENT can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell. See *Defining Zone-Based Reaction Mechanisms* (p. 816) for details about defining reaction mechanisms. See *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829) for details about wall surface reactions.

7.2.3.6.5. Including the Relative Velocity Resistance Formulation

Prior to ANSYS FLUENT 6.3, cases with moving reference frames used the absolute velocities in the source calculations for inertial and viscous resistance. This approach has been enhanced so that relative velocities are used for the porous source calculations (*Momentum Equations for Porous Media* (p. 222)). Using the **Relative Velocity Resistance Formulation** option (turned on by default) allows you to better predict the source terms for cases involving moving meshes or moving reference frames (MRF). This option works well in cases with non-moving and moving porous media. Note that ANSYS FLUENT will use the appropriate velocities (relative or absolute), depending on your case setup.

7.2.3.6.6. Defining the Viscous and Inertial Resistance Coefficients

The viscous and inertial resistance coefficients are both defined in the same manner. The basic approach for defining the coefficients using a Cartesian coordinate system is to define one direction vector in 2D or two direction vectors in 3D, and then specify the viscous and/or inertial resistance coefficients in each direction. In 2D, the second direction, which is not explicitly defined, is normal to the plane defined by the specified direction vector and the z direction vector. In 3D, the third direction is normal to the plane defined by the two specified direction vectors. For a 3D problem, the second direction must be normal to the first. If you fail to specify two normal directions, the solver will ensure that they are normal by ignoring any component of the second direction that is in the first direction. You should therefore be certain that the first direction is correctly specified.

You can also define the viscous and/or inertial resistance coefficients in each direction using a user-defined function (UDF). The user-defined options become available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS FLUENT. Note that the coefficients defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined function, see the UDF Manual.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viscous and/or inertial resistance coefficients can be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

If you are modeling axisymmetric swirling flows, you can specify an additional direction component for the viscous and/or inertial resistance coefficients. This direction component is always tangential to the other two specified directions. This option is available for both density-based and pressure-based solvers.

In 3D, it is also possible to define the coefficients using a conical (or cylindrical) coordinate system, as described below.

Important

Note that the viscous and inertial resistance coefficients are generally based on the superficial velocity of the fluid in the porous media.

The procedure for defining resistance coefficients is as follows:

1. Define the direction vectors.
   - To use a Cartesian coordinate system, simply specify the **Direction-1 Vector** and, for 3D, the **Direction-2 Vector**. The unspecified direction will be determined as described above. These direction vectors correspond to the principle axes of the porous media.

     Note

     The units for the inertial resistance coefficients (**Direction-1 Vector** and **Direction-2 Vector**) is the inverse of length. Should you want to define different units, you can do so by opening the **Units** dialog box and selecting **resistance** from the **Quantities** list.

     For some problems in which the principal axes of the porous medium are not aligned with the coordinate axes of the domain, you may not know a priori the direction vectors of the porous medium. In such cases, the plane tool in 3D (or the line tool in 2D) can help you to determine these direction vectors.

     a. "Snap" the plane tool (or the line tool) onto the boundary of the porous region. (Follow the instructions in *Using the Plane Tool* (p. 1367) or *Using the Line Tool* (p. 1363) for initializing the tool to a position on an existing surface.)

     b. Rotate the axes of the tool appropriately until they are aligned with the porous medium.

     c. Once the axes are aligned, click on the **Update From Plane Tool** or **Update From Line Tool** button in the **Fluid** dialog box. ANSYS FLUENT will automatically set the **Direction-1 Vector** to the direction of the red arrow of the tool, and (in 3D) the **Direction-2 Vector** to the direction of the green arrow.
   - To use a conical coordinate system (e.g., for an annular, conical filter element), follow the steps below. This option is available only in 3D cases.

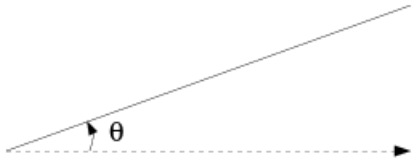
     a. Turn on the **Conical** option.

     b. Set the **Cone Half Angle** (the angle between the cone's axis and its surface, shown in *Figure 7.9* (p. 230)). To use a cylindrical coordinate system, set the **Cone Half Angle** to 0.

     c. Specify the **Cone Axis Vector** and **Point on Cone Axis**. The cone axis is specified as being in the direction of the **Cone Axis Vector** (unit vector), and passing through the **Point on Cone Axis**. The cone axis may or may not pass through the origin of the coordinate system.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.9 Cone Half Angle



For some problems in which the axis of the conical filter element is not aligned with the coordinate axes of the domain, you may not know a priori the direction vector of the cone axis and coordinates of a point on the cone axis. In such cases, the plane tool can help you to determine the cone axis vector and point coordinates. One method is as follows:

a. Select a boundary zone of the conical filter element that is normal to the cone axis vector in the drop-down list next to the **Snap to Zone** button.

b. Click the **Snap to Zone** button. ANSYS FLUENT will automatically "snap" the plane tool onto the boundary. It will also set the **Cone Axis Vector** and the **Point on Cone Axis**. (Note that you will still have to set the **Cone Half Angle** yourself.)

An alternate method is as follows:

a. "Snap" the plane tool onto the boundary of the porous region. (Follow the instructions in *Using the Plane Tool* (p. 1367) for initializing the tool to a position on an existing surface.)

b. Rotate and translate the axes of the tool appropriately until the red arrow of the tool is pointing in the direction of the cone axis vector and the origin of the tool is on the cone axis.

c. Once the axes and origin of the tool are aligned, click the **Update From Plane Tool** button in the **Fluid** dialog box. ANSYS FLUENT will automatically set the **Cone Axis Vector** and the **Point on Cone Axis**. (Note that you will still have to set the **Cone Half Angle** yourself.)

2. Under **Viscous Resistance**, specify the viscous resistance coefficient $1/\alpha$ in each direction.

   Under **Inertial Resistance**, specify the inertial resistance coefficient C_2 in each direction. (You will need to scroll down with the scroll bar to view these inputs.)

   For porous media cases containing highly anisotropic inertial resistances, enable **Alternative Formulation** under **Inertial Resistance**. The **Alternative Formulation** option provides better stability to the calculation when your porous medium is anisotropic. The pressure loss through the medium depends on the magnitude of the velocity vector of the ith component in the medium. Using the formulation of *Equation 7–6* (p. 224) yields the expression below:

$$S_i = \frac{1}{2}\rho C_i |v_i| v_i \qquad \textbf{(7–10)}$$

   Whether or not you use the **Alternative Formulation** option depends on how well you can fit your experimentally determined pressure drop data to the ANSYS FLUENT model. For example, if the flow through the medium is aligned with the mesh in your ANSYS FLUENT model, then it will not make a difference whether or not you use the formulation.

   For more information about simulations involving highly anisotropic porous media, see *Solution Strategies for Porous Media* (p. 240).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the alternative formulation is compatible only with the pressure-based solver.

If you are using the **Conical** specification method, **Direction-1** is the tangential direction of the cone, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) direction.

In 3D there are three possible categories of coefficients, and in 2D there are two:

- In the isotropic case, the resistance coefficients in all directions are the same (e.g., a sponge). For an isotropic case, you must explicitly set the resistance coefficients in each direction to the same value.
- When (in 3D) the coefficients in two directions are the same and those in the third direction are different or (in 2D) the coefficients in the two directions are different, you must be careful to specify the coefficients properly for each direction. For example, if you had a porous region consisting of cylindrical straws with small holes in them positioned parallel to the flow direction, the flow would pass easily through the straws, but the flow in the other two directions (through the small holes) would be very little. If you had a plane of flat plates perpendicular to the flow direction, the flow would not pass through them at all; it would instead move in the other two directions.
- In 3D the third possible case is one in which all three coefficients are different. For example, if the porous region consisted of a plane of irregularly-spaced objects (e.g., pins), the movement of flow between the blockages would be different in each direction. You would therefore need to specify different coefficients in each direction.

Methods for deriving viscous and inertial loss coefficients are described in the sections that follow.

7.2.3.6.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss

When you use the porous media model, you must keep in mind that the porous cells in ANSYS FLUENT are *100% open*, and that the values that you specify for $1/\alpha_{ij}$ and/or $C_{2_{ij}}$ must be based on this assumption. Suppose, however, that you know how the pressure drop varies with the velocity through the actual device, which is only partially open to flow. The following exercise is designed to show you how to compute a value for C_2 which is appropriate for the ANSYS FLUENT model.

Consider a perforated plate which has 25% area open to flow. The pressure drop through the plate is known to be 0.5 times the dynamic head in the plate. The loss factor, K_L, defined as

$$\Delta p = K_L \left(\frac{1}{2} \rho v_{25\% open}^2 \right) \tag{7–11}$$

is therefore 0.5, based on the actual fluid velocity in the plate, i.e., the velocity through the 25% open area. To compute an appropriate value for C_2, note that in the ANSYS FLUENT model:

1. The velocity through the perforated plate assumes that the plate is 100% open.
2. The loss coefficient must be converted into dynamic head loss per unit length of the porous region.

Noting item 1, the first step is to compute an adjusted loss factor, K_L', which would be based on the velocity of a 100% open area:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta p = K_L' \left(\frac{1}{2} \rho v_{100\%open}^2 \right) \tag{7–12}$$

or, noting that for the same flow rate, $v_{25\%open} = 4 \times v_{100\%open}$,

$$\begin{aligned} K_L' &= K_L \times \frac{v_{25\%open}^2}{v_{100\%open}^2} \\ &= 0.5 \times \left(\frac{4}{1} \right)^2 \\ &= 8 \end{aligned} \tag{7–13}$$

The adjusted loss factor has a value of 8. Noting item 2, you must now convert this into a loss coefficient per unit thickness of the perforated plate. Assume that the plate has a thickness of 1.0 mm (10^{-3} m). The inertial loss factor would then be

$$C_2 = \frac{K_L'}{thickness} = \frac{8}{10^{-3}} = 8000 m^{-1} \tag{7–14}$$

Note that, for anisotropic media, this information must be computed for each of the 2 (or 3) coordinate directions.

7.2.3.6.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed

As a second example, consider the modeling of a packed bed. In turbulent flows, packed beds are modeled using both a permeability and an inertial loss coefficient. One technique for deriving the appropriate constants involves the use of the Ergun equation [*21*] (p. 2210), a semi-empirical correlation applicable over a wide range of Reynolds numbers and for many types of packing:

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2} \frac{(1-\varepsilon)^2}{\varepsilon^3} v_\infty + \frac{1.75\rho}{D_p} \frac{(1-\varepsilon)}{\varepsilon^3} v_\infty^2 \tag{7–15}$$

When modeling laminar flow through a packed bed, the second term in the above equation may be dropped, resulting in the Blake-Kozeny equation [*21*] (p. 2210):

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2} \frac{(1-\varepsilon)^2}{\varepsilon^3} v_\infty \tag{7–16}$$

In these equations, μ is the viscosity, D_p is the mean particle diameter, L is the bed depth, and ε is the void fraction, defined as the volume of voids divided by the volume of the packed bed region. Comparing *Equation 7–4* (p. 223) and *Equation 7–6* (p. 224) with *Equation 7–15* (p. 232), the permeability and inertial loss coefficient in each component direction may be identified as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\alpha = \frac{D_p^2}{150} \frac{\varepsilon^3}{(1-\varepsilon)^2} \tag{7–17}$$

and

$$C_2 = \frac{3.5}{D_p} \frac{(1-\varepsilon)}{\varepsilon^3} \tag{7–18}$$

7.2.3.6.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate

As a third example we will take the equation of Van Winkle et al. [*63*] (p. 2212) [*83*] (p. 2213) and show how porous media inputs can be calculated for pressure loss through a perforated plate with square-edged holes.

The expression, which is claimed by the authors to apply for turbulent flow through square-edged holes on an equilateral triangular spacing, is

$$\dot{m} = C A_f \sqrt{(2\rho \Delta p) / \left(1 - \left(A_f / A_p\right)^2\right)} \tag{7–19}$$

where

$\dot{m}$ = mass flow rate through the plate

A_f = the free area or total area of the holes

A_p = the area of the plate (solid and holes)

C = a coefficient that has been tabulated for various Reynolds-number ranges and for various D/t

D/t = the ratio of hole diameter to plate thickness

for $t/D > 1.6$ and for $Re > 4000$ the coefficient C takes a value of approximately 0.98, where the Reynolds number is based on hole diameter and velocity in the holes.

Rearranging *Equation 7–19* (p. 233), making use of the relationship

$$\dot{m} = \rho v A_p \tag{7–20}$$

and dividing by the plate thickness, $\Delta x = t$, we obtain

$$\frac{\Delta p}{\Delta x} = \left(\frac{1}{2}\rho v^2\right) \frac{1}{C^2} \frac{\left(A_p / A_f\right)^2 - 1}{t} \tag{7–21}$$

where v is the superficial velocity (not the velocity in the holes). Comparing with *Equation 7–6* (p. 224) is seen that, for the direction normal to the plate, the constant C_2 can be calculated from

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$C_2 = \frac{1}{C^2} \frac{\left(A_p/A_f\right)^2 - 1}{t} \tag{7–22}$$

7.2.3.6.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat

Consider the problem of laminar flow through a mat or filter pad which is made up of randomly-oriented fibers of glass wool. As an alternative to the Blake-Kozeny equation (*Equation 7–16* (p. 232)) we might choose to employ tabulated experimental data. Such data is available for many types of fiber [*36*] (p. 2210).

volume fraction of solid material	dimensionless permeability B of glass wool
0.262	0.25
0.258	0.26
0.221	0.40
0.218	0.41
0.172	0.80

where $B = \alpha / a^2$ and a is the fiber diameter. α, for use in *Equation 7–4* (p. 223), is easily computed for a given fiber diameter and volume fraction.

7.2.3.6.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data

Experimental data that is available in the form of pressure drop against velocity through the porous component, can be extrapolated to determine the coefficients for the porous media. To effect a pressure drop across a porous medium of thickness, Δn, the coefficients of the porous media are determined in the manner described below.

If the experimental data is:

Velocity (m/s)	Pressure Drop (Pa)
20.0	28.4
50.0	487.0
80.0	1432.0
110.0	2964.0

then an xy curve can be plotted to create a trendline through these points yielding the following equation

$$\Delta p = 0.28296 v^2 - 4.33539 v \tag{7–23}$$

where Δp is the pressure drop and v is the velocity.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Although the best fit curve may yield negative coefficients, it should be avoided when using the porous media model in ANSYS FLUENT.

Note that a simplified version of the momentum equation, relating the pressure drop to the source term, can be expressed as

$$\nabla p = S_i \tag{7–24}$$

or

$$\Delta p = -S_i \Delta n \tag{7–25}$$

Hence, comparing *Equation 7–23* (p. 234) to *Equation 7–2* (p. 222), yields the following curve coefficients:

$$0.28296 = C_2 \frac{1}{2} \rho \Delta n \tag{7–26}$$

with $\rho = 1.225$ kg/m^3, and a porous media thickness, Δn, assumed to be 1m in this example, the inertial resistance factor, $C_2 = 0.462$.

Likewise,

$$-4.33539 = \frac{\mu}{\alpha} \Delta n \tag{7–27}$$

with $\mu = 1.7894 \times 10^{-5}$, the viscous inertial resistance factor, $\frac{1}{\alpha} = -242282$.

Important

Note that this same technique can be applied to the porous jump boundary condition. Similar to the case of the porous media, you have to take into account the thickness of the medium Δn. Your experimental data can be plotted in an xy curve, yielding an equation that is equivalent to *Equation 7–112* (p. 337). From there, you can determine the permeability α and the pressure jump coefficient C_2.

7.2.3.6.12. Using the Power-Law Model

If you choose to use the power-law approximation of the porous-media momentum source term (*Equation 7–3* (p. 223)), the only inputs required are the coefficients C_0 and C_1. Under **Power Law Model** in the **Fluid** dialog box, enter the values for **C0** and **C1**. Note that the power-law model can be used in conjunction with the Darcy and inertia models.

C0 must be in SI units, consistent with the value of **C1**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.6.13. Defining Porosity

To define the porosity, scroll down below the resistance inputs in the **Fluid** dialog box, and set the **Porosity** under **Fluid Porosity**.

You can also define the porosity using a user-defined function (UDF). The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS FLUENT. Note that the porosity defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined function, see the separate UDF Manual.

Porosity can be assigned to an input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

The porosity, γ, is the volume fraction of fluid within the porous region (i.e., the open volume fraction of the medium). The porosity is used in the prediction of heat transfer in the medium, as described in *Treatment of the Energy Equation in Porous Media* (p. 224), and in the time-derivative term in the scalar transport equations for unsteady flow, as described in *Effect of Porosity on Transient Scalar Equations* (p. 225). It also impacts the calculation of reaction source terms and body forces in the medium. These sources will be proportional to the fluid volume in the medium. If you want to represent the medium as completely open (no effect of the solid medium), you should set the porosity equal to 1.0 (the default). When the porosity is equal to 1.0, the solid portion of the medium will have no impact on heat transfer or thermal/reaction source terms in the medium.

7.2.3.6.14. Defining the Porous Material

If you choose to model heat transfer in the porous medium, you must specify the material contained in the porous medium.

To define the material contained in the porous medium, scroll down below the resistance inputs in the **Fluid** dialog box, and select the appropriate solid in the **Solid Material Name** drop-down list under **Fluid Porosity**. If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box. In the **Edit Material** dialog box, you can define the non-isotropic thermal conductivity of the porous material using a user-defined function (UDF). The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS FLUENT. Note that the non-isotropic thermal conductivity defined in the UDF must utilize the `DEFINE_PROPERTY` macro. For more information on creating and using user-defined function, see the separate UDF Manual.

7.2.3.6.15. Defining Sources

If you want to include effects of the heat generated by the porous medium in the energy equation, enable the **Source Terms** option and set a non-zero **Energy** source. The solver will compute the heat generated by the porous region by multiplying this value by the total volume of the cells comprising the porous zone. You may also define sources of mass, momentum, turbulence, species, or other scalar quantities, as described in *Defining Mass, Momentum, Energy, and Other Sources* (p. 244).

7.2.3.6.16. Defining Fixed Values

If you want to fix the value of one or more variables in the fluid region of the zone, rather than computing them during the calculation, you can do so by enabling the **Fixed Values** option. See *Fixing the Values of Variables* (p. 241) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.6.17. Suppressing the Turbulent Viscosity in the Porous Region

As discussed in *Treatment of Turbulence in Porous Media* (p. 225), turbulence will be computed in the porous region just as in the bulk fluid flow. If you are using one of the and you want the turbulence generation to be zero in the porous zone, turn on the **Laminar Zone** option in the **Fluid** dialog box. Refer to *Specifying a Laminar Zone* (p. 216) for more information about suppressing turbulence generation.

7.2.3.6.18. Specifying the Rotation Axis and Defining Zone Motion

Inputs for the rotation axis and zone motion are the same as for a standard fluid zone. See *Inputs for Fluid Zones* (p. 215) for details.

7.2.3.7. Modeling Porous Media Based on Physical Velocity

As stated in *Limitations and Assumptions of the Porous Media Model* (p. 221), by default ANSYS FLUENT calculates the superficial velocity based on volumetric flow rate. The superficial velocity in the governing equations can be represented as

$$\vec{v}_{superficial} = \gamma \vec{v}_{physical} \qquad \textbf{(7–28)}$$

where γ is the porosity of the media defined as the ratio of the volume occupied by the fluid to the total volume.

The superficial velocity values within the porous region remain the same as those outside of the porous region, and porosity is not taken into account in the differential terms of the transport equations. This limits the accuracy of the porous model in cases where there should be an increase in velocity throughout the porous region, and does not yield accurate results when velocity values and gradients are important. For more accurate simulations of porous media flows, it becomes necessary to solve for the true, or physical, velocity throughout the flowfield rather than the superficial velocity, as well as to include porosity in all terms of the transport equations.

ANSYS FLUENT allows the calculation of the physical velocity using the **Porous Formulation**, available in the **Cell Zone Conditions** task page. By default, the **Superficial Velocity** option is turned on.

7.2.3.7.1. Single Phase Porous Media

Using the physical velocity formulation, and assuming a general scalar ϕ, the governing equation in an isotropic porous media has the following form:

$$\frac{\partial(\gamma\rho\phi)}{\partial t} + \nabla\cdot(\gamma\rho\vec{v}\phi) = \nabla\cdot(\gamma\Gamma\nabla\phi) + \gamma S_\phi \qquad \textbf{(7–29)}$$

Assuming isotropic porosity and single phase flow, the volume-averaged mass and momentum conservation equations are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial(\gamma\rho)}{\partial t}+\nabla\cdot(\gamma\rho\vec{v})=0 \tag{7–30}$$

$$\frac{\partial(\gamma\rho\vec{v})}{\partial t}+\nabla\cdot(\gamma\rho\vec{v}\,\vec{v})=-\gamma\nabla p+\nabla\cdot(\gamma\vec{\tau})+\gamma\vec{B}_f-\left(\frac{\mu}{\alpha}+\frac{C_2\rho}{2}|\vec{v}|\right)\vec{v} \tag{7–31}$$

The last term in *Equation 7–31* (p. 238) represents the viscous and inertial drag forces imposed by the pore walls on the fluid.

Important

Note that even when you solve for the physical velocity in *Equation 7–31* (p. 238), the two resistance coefficients can still be derived using the superficial velocity as given in *Defining the Viscous and Inertial Resistance Coefficients* (p. 228). ANSYS FLUENT assumes that the inputs for these resistance coefficients are based upon well-established empirical correlations that are usually based on superficial velocity. Therefore, ANSYS FLUENT automatically converts the inputs for the resistance coefficients into those that are compatible with the physical velocity formulation.

Important

Note that the inlet mass flow is also calculated from the superficial velocity. Therefore, for the same mass flow rate at the inlet and the same resistance coefficients, for either the physical or superficial velocity formulation you should obtain the same pressure drop across the porous media zone.

7.2.3.7.2. Multiphase Porous Media

You can simulate porous media multiphase flows using the **Physical Velocity Porous Formulation** to solve the true or physical velocity field throughout the entire flow field, including both porous and non-porous regions. In this approach, assuming a general scalar in the qth phase, ϕ_q, the governing equation in an isotropic porous medium takes on the following form:

$$\frac{\partial\left(\gamma\alpha_q\rho_q\phi_q\right)}{\partial t}+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\phi_q\right)=\nabla\cdot\left(\gamma\Gamma_q\nabla\phi_q\right)+\gamma S_{\phi,q} \tag{7–32}$$

Here γ is the porosity, which may vary with time and space; ρ_q is the phase density; α_q is the volume fraction; $\vec{v}_q$ is the phase velocity vector; $S_{\phi,q}$ is the source term; and Γ_q is the diffusion coefficient.

The general scalar equation *Equation 7–32* (p. 238) applies to all other transport equations in the Eulerian multiphase model, such as the granular phase momentum and energy equations, turbulence modeling equations, and the species transport equations.

Assuming isotropic porosity and multiphase flows, the governing equations in the qth phase, Equation 17–119, Equation 17–120, and Equation 17–126 in the Theory Guide take the general forms described below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.7.2.1. The Continuity Equation

$$\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\right)=\gamma\sum_{p=1}^{n}\left(\dot{m}_{pq}-\dot{m}_{qp}\right)+\gamma S_q \tag{7–33}$$

7.2.3.7.2.2. The Momentum Equation

$$\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q\vec{v}_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\vec{v}_q\right)=-\gamma\alpha_q\nabla p+\nabla\cdot\left(\gamma\overline{\overline{\tau}}_q\right)+\gamma\alpha_q\rho_q\vec{g}+ \tag{7–34}$$

$$\gamma\sum_{p=1}^{n}\left(\vec{R}_{pq}+\dot{m}_{pq}\vec{v}_{pq}-\dot{m}_{qp}\vec{v}_{qp}\right)+\gamma\left(\vec{F}_q+\vec{F}_{lift,q}+\vec{F}_{vm,q}\right)+\alpha_q\left(\frac{\mu}{K}+\frac{C_2\rho}{2}|\vec{v}_q|\right)\vec{v}_q$$

where the last term in *Equation 7–34* (p. 239) is the momentum resistance (sink) source in a porous medium. It consists of two parts: a viscous loss term, and an inertial loss term. The parameter K is the permeability, and C_2 is the inertial resistance factor. Both K and C_2 are functions of $(1-\gamma)$. When $\gamma=1$, the flow is non-porous and the two loss terms disappear. Details about the user inputs related to the momentum resistance sources can be found in *User Inputs for Porous Media* (p. 225).

7.2.3.7.2.3. The Energy Equation

$$\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q h_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q h_q\right)=-\gamma\alpha_q\frac{\partial p_q}{\partial t}+\gamma\overline{\overline{\tau}}_q:\nabla\vec{v}_q-\nabla\cdot\left(\gamma\vec{q}_q\right)+\gamma S_q+ \tag{7–35}$$

$$\gamma\sum_{p=1}^{n}\left(Q_{pq}+\dot{m}_{pq}h_{pq}-\dot{m}_{qp}h_{qp}\right)+Q_{sp}$$

where Q_{sp} is the heat transfer between the solids surface and the phase q in a porous medium. Assuming only convective heat transfer, we then have

$$Q_{sp}=(1-\gamma)\alpha_q h_{q,eff}\left(T_s-T_q\right) \tag{7–36}$$

where $h_{q,eff}$ is the effective convective heat transfer coefficient, and T_s is the solids surface temperature in the porous medium. It is governed by the heat conduction equation:

$$\frac{\partial}{\partial t}\left(\rho_s h_s\right)+\nabla\cdot\left(\vec{v}_s\rho_s h_s\right)=\nabla\cdot(k_s\nabla T)-\sum_{p=1}^{n}Q_{sp} \tag{7–37}$$

Equation 7–37 (p. 239) can be solved as a user-defined scalar (UDS) equation, as described in *User-Defined Scalar (UDS) Transport Equations* (p. 485). By default, ANSYS FLUENT assumes that the overall heat transfer between the multiphase fluid and the solids is in equilibrium. Therefore, instead of solving *Equation 7–37* (p. 239) to obtain the solids surface temperature, we have

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\sum_{p=1}^{n} Q_{sp} = 0 \tag{7–38}$$

and then

$$T_s = \frac{\sum_{p=1}^{n} \alpha_p h_{p,eff} T_p}{\sum_{p=1}^{n} \alpha_p h_{p,eff}} \tag{7–39}$$

7.2.3.8. Solution Strategies for Porous Media

In general, you can use the standard solution procedures and solution parameter settings when your ANSYS FLUENT model includes porous media. You may find, however, that the rate of convergence slows when you define a porous region through which the pressure drop is relatively large in the flow direction (e.g., the permeability, α, is low or the inertial factor, C_2, is large). This slow convergence can occur because the porous media pressure drop appears as a momentum source term—yielding a loss of diagonal dominance—in the matrix of equations solved. The best remedy for poor convergence of a problem involving a porous medium is to supply a good initial guess for the pressure drop across the medium. You can supply this guess by patching a value for the pressure in the fluid cells upstream and/or downstream of the medium, as described in *Patching Values in Selected Cells* (p. 1242). It is important to recall, when patching the pressure, that the pressures you input should be defined as the gauge pressures used by the solver (i.e., relative to the operating pressure defined in the *Operating Conditions Dialog Box* (p. 1806)).

Another possible way to deal with poor convergence is to temporarily disable the porous media model (by turning off the **Porous Zone** option in the *Fluid Dialog Box* (p. 1796)) and obtain an initial flow field without the effect of the porous region. With the porous media model turned off, ANSYS FLUENT will treat the porous zone as a fluid zone and calculate the flow field accordingly. Once an initial solution is obtained, or the calculation is proceeding steadily to convergence, you can enable the porous media model and continue the calculation with the porous region included. (This method is not recommended for porous media with high resistance.)

Simulations involving highly anisotropic porous media may, at times, pose convergence troubles. You can address these issues by limiting the anisotropy of the porous media coefficients ($1/\alpha_{ij}$ and $C_{2_{i,j}}$) to two or three orders of magnitude. Even if the medium's resistance in one direction is infinite, you do not need to set the resistance in that direction to be greater than 1000 times the resistance in the primary flow direction.

7.2.3.9. Postprocessing for Porous Media

The impact of a porous region on the flow field can be determined by examining either velocity components or pressure values. Graphical plots (including XY plots and contour or vector plots) or alphanumeric reports of the following variables/functions may be of interest:

- **X**, **Y**, **Z Velocity** (in the **Velocity...** category)
- **Static Pressure** (in the **Pressure...** category)

These variables are contained in the specified categories of the variable selection drop-down list that appears in postprocessing dialog boxes.

Note that thermal reporting in the porous region is defined as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k_{eff} = \gamma k_s + (1-\gamma) k_f \quad \textbf{(7–40)}$$

where

γ = porosity of the medium

k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_s = solid medium thermal conductivity

Important

For porous media involving surface reactions, you can display/report the surface reaction rates using the **Arrhenius Rate of Reaction-n** in the **Reactions...** category of the variable selection drop-down list.

7.2.4. Fixing the Values of Variables

The option to fix values of variables in ANSYS FLUENT allows you to set the value of one or more variables in a fluid or solid zone, essentially setting a boundary condition for the variables within the cells of the zone. When a variable is fixed in a given cell, the transport equation for that variable is not solved in the cell (and the cell is not included when the residual sum is computed for that variable). The fixed value is used for the calculation of face fluxes between the cell and its neighbors. The result is a smooth transition between the fixed value of a variable and the values at the neighboring cells.

Important

You can fix values for temperature and species mass fractions only if you are using the pressure-based solver. You can fix values for velocity components only if you are using the pressure-based segregated solver. (Refer to Pressure-Based Solver in the Theory Guide for information about the pressure-based segregated solver.)

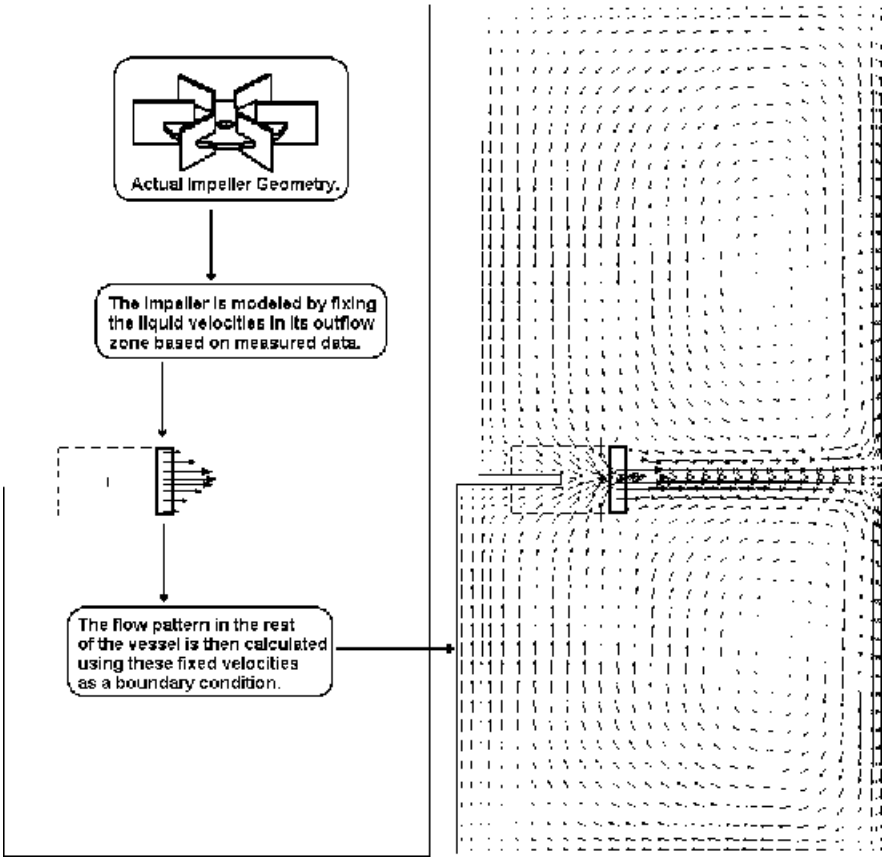
7.2.4.1. Overview of Fixing the Value of a Variable

The ability to fix the value of a variable has a wide range of applications. The velocity fixing method is often used to model the flow in stirred tanks. This approach provides an alternative to the use of a moving reference frame (solution in the reference frame of the blade) and can be used to model baffled tanks. In both 2D and 3D geometries, a fluid cell zone may be used in the impeller regions, and velocity components can be fixed based on measured data.

Although the actual impeller geometry can be modeled and the flow pattern calculated using the sliding mesh model, experimental data for the velocity profile in the outflow region are available for many impeller types. If you do not need to know the details of the flow around the blades for your problem, you can model the impeller by fixing the experimentally-obtained liquid velocities in its outflow zone. The velocities in the rest of the vessel can then be calculated using this fixed velocity profile as a boundary condition. *Figure 7.10* (p. 242) shows an example of how this method is used to model the flow pattern created by a disk-turbine in an axisymmetric stirred vessel.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.10 Fixing Values for the Flow in a Stirred Tank



7.2.4.1.1. Variables That Can Be Fixed

The variables that can be fixed include velocity components (pressure-based segregated solver only), turbulence quantities, temperature (pressure-based solver only), enthalpy, species mass fractions (pressure-based solver only), and user-defined scalars. For turbulence quantities, different values can be set depending on your choice of turbulence model. You can fix the value of the temperature in a fluid or solid zone if you are solving the energy equation. If you are using the non-premixed combustion model, you can fix the enthalpy in a fluid zone. If you have more than one species in your model, you can specify fixed values for the species mass fractions for each individual species except the last one you defined. See the UDF Manual for details about defining user-defined scalars.

If you are using the Eulerian multiphase model, you can fix the values of velocity components and (depending on which multiphase turbulence model you are using) turbulence quantities on a per-phase basis. See *Eulerian Model* (p. 1111) for details about setting boundary conditions for Eulerian multiphase calculations.

7.2.4.2. Procedure for Fixing Values of Variables in a Zone

To fix the values of one or more variables in a cell zone, follow these steps (remembering to use only SI units):

1. In the *Fluid Dialog Box* (p. 1796) or *Solid Dialog Box* (p. 1802), turn on the **Fixed Values** option.
2. Fix the values for the appropriate variables, noting the comments below.
   - To specify a constant value for a variable, choose **constant** in the drop-down list next to the relevant field and then enter the constant value in the field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To specify a non-constant value for a variable, you can use a profile (see *Profiles* (p. 366)) or a user-defined function for a profile (see the UDF Manual). Select the appropriate profile or UDF in the drop-down list next to the relevant field.

  If you specify a radial-type profile (see *Profile Specification Types* (p. 366)) for temperature, enthalpy, species mass/mole fractions, or turbulence quantities for the k- ε, Spalart-Allmaras, or k- ω model, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See *Specifying the Rotation Axis* (p. 217) for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)
- If you do not want to fix the value for a variable, choose (or keep) **none** in the drop-down list next to the relevant field. This is the default for all variables.

7.2.4.2.1. Fixing Velocity Components

To fix the velocity components, you can specify **X**, **Y**, and (in 3D) **Z Velocity** values, or, for axisymmetric cases, **Axial**, **Radial**, and (for axisymmetric swirl) **Swirl Velocity** values. The units for a fixed velocity are m/s.

For 3D cases, you can choose to specify cylindrical velocity components instead of Cartesian components. Turn on the **Local Coordinate System For Fixed Velocities** option, and then specify the **Axial**, **Radial**, and/or **Tangential Velocity** values. The local coordinate system is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See *Specifying the Rotation Axis* (p. 217) for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

Important

You can fix values for velocity components only if you are using the pressure-based segregated solver. (Refer to Pressure-Based Solver in the Theory Guide for information about the pressure-based segregated solver.)

7.2.4.2.2. Fixing Temperature and Enthalpy

If you are solving the energy equation, you can fix the temperature in a zone by specifying the value of the **Temperature**. The units for a fixed temperature are K.

If you are using the non-premixed combustion model, you can fix the enthalpy in a zone by specifying the value of the **Enthalpy**. The units for a fixed enthalpy are J/kg.

If you specify a radial-type profile (see *Profile Specification Types* (p. 366)) for temperature or enthalpy, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details.

Important

You can fix the value of temperature only if you are using the pressure-based solver.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.4.2.3. Fixing Species Mass Fractions

If you are using the species transport model, you can fix the values of the species mass fractions for individual species. ANSYS FLUENT allows you to fix the species mass fraction for each species (e.g., **h2**, **o2**) except the last one you defined.

If you specify a radial-type profile (see *Profile Specification Types* (p. 366)) for a species mass fraction, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details.

Important

You can fix values for species mass fractions only if you are using the pressure-based solver.

7.2.4.2.4. Fixing Turbulence Quantities

To fix the values of k and ε in the k- ε equations, specify the **Turbulence Kinetic Energy** and **Turbulence Dissipation Rate** values. The units for k are m^2/s^2 and those for ε are m^2/s^3.

To fix the value of the modified turbulent viscosity ($\tilde{\nu}$) for the Spalart-Allmaras model, specify the **Modified Turbulent Viscosity** value. The units for the modified turbulent viscosity are m^2/s.

To fix the values of k and ω in the k- ω equations, specify the **Turbulence Kinetic Energy** and **Specific Dissipation Rate** values. The units for k are m^2/s^2 and those for ω are 1/s.

To fix the value of k, ε, or the Reynolds stresses in the RSM transport equations, specify the **Turbulence Kinetic Energy**, **Turbulence Dissipation Rate**, **UU Reynolds Stress**, **VV Reynolds Stress**, **WW Reynolds Stress**, **UV Reynolds Stress**, **VW Reynolds Stress**, and/or **UW Reynolds Stress**. The units for k and the Reynolds stresses are m^2/s^2, and those for ε are m^2/s^3.

If you specify a radial-type profile (see *Profile Specification Types* (p. 366)) for k, ε, ω, or $\tilde{\nu}$, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details. Note that you cannot specify radial-type profiles for the Reynolds stresses.

7.2.4.2.5. Fixing User-Defined Scalars

To fix the value of a user-defined scalar, specify the **User defined scalar-n** value. (There will be one for each user-defined scalar you have defined.) The units for a user-defined scalar will be the appropriate SI units for the scalar quantity. See the UDF Manual for information on user-defined scalars.

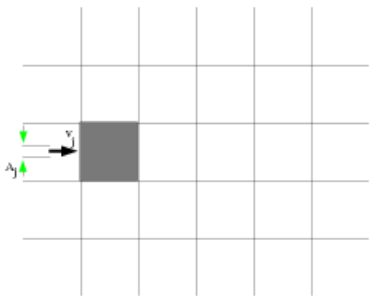
7.2.5. Defining Mass, Momentum, Energy, and Other Sources

You can define volumetric sources of mass (for single or multiple species), momentum, energy, turbulence, and other scalar quantities in a fluid zone, or a source of energy for a solid zone. This feature is useful when you want to input a known value for these sources. (For more complicated sources with functional dependency, you can create a user-defined function as described in the separate UDF Manual.) To add source terms to a cell or group of cells, you must place the cell(s) in a separate zone. The sources are then applied to that cell zone. Typical uses for this feature are listed below:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A flow source that cannot be represented by an inlet, e.g., due to an issue of scale. If you need to model an inlet that is smaller than a cell, you can place the cell where the tiny "inlet" is located in its own fluid zone and then define the mass, momentum, and energy sources in that cell zone. For the example shown in *Figure 7.11* (p. 245), you should set a mass source of $\frac{\dot{m}}{V} = \frac{\rho_j A_j v_j}{V}$ and a momentum source of $\frac{\dot{m}v}{V} = \frac{\dot{m}v_j}{V}$, where V is the cell volume.
- Heat release due to a source (e.g., fire) that is not explicitly defined in your model. For this case, you can place the cell(s) into which the heat is originally released in its own fluid zone and then define the energy source in that cell zone.
- An energy source in a solid zone, for conjugate heat transfer applications. For this case, you can place the cell(s) into which the heat is originally released in its own solid zone and then define the energy source in that cell zone.
- A species source due to a reaction that is not explicitly included in the model. In the above example of simulating a fire, you might need to define a source for a species representing smoke generation.

Figure 7.11 Defining a Source for a Tiny Inlet



Important

Note that if you define a mass source for a cell zone, you should also define a momentum source and, if appropriate for your model, energy and turbulence sources. If you define only a mass source, that mass enters the domain with no momentum or thermal heat. The mass will therefore have to be accelerated and heated by the flow and, consequently, there may be a drop in velocity or temperature. This drop may or may not be perceptible, depending on the size of the source. (Note that defining only a momentum, energy, or turbulence source is acceptable.)

7.2.5.1. Sign Conventions and Units

All positive source terms indicate sources, and all negative source terms indicate sinks. All sources must be specified in SI units.

7.2.5.2. Procedure for Defining Sources

To define one or more source terms for a zone, follow these steps (remembering to use only SI units):

1. In the *Fluid Dialog Box* (p. 1796) or *Solid Dialog Box* (p. 1802), turn on the **Source Terms** option.
2. Set the appropriate source terms under the **Source Terms** tab, noting the comments below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To specify a source, click the **Edit...** button next to the mass, momentum, energy, or other source. The sources dialog box will open where you will define the number of sources. For each source, choose **constant**, **user-defined**, or **none** in the drop-down list.
- To specify a constant source, choose **constant** in the drop-down list and then enter the constant value in the field.
- To specify a temperature-dependent or other functional source, you can use a user-defined function (see the UDF Manual).
- If you do not want to specify a source term for a variable, choose (or keep) **none** in the drop-down list next to the relevant field. This is the default for all variables.
- Remember that you should not define just a mass source without defining the other sources, as described in *Defining Mass, Momentum, Energy, and Other Sources* (p. 244). above.
- Since the sources you specify are defined per unit volume, to determine the appropriate value of your source term you will often need to first determine the volume of the cell(s) in the zone for which you are defining the source. To do this, you can use the *Volume Integrals Dialog Box* (p. 2034).

7.2.5.2.1. Mass Sources

If you have only one species in your problem, you can simply define a **Mass** source for that species. The units for the mass source are kg/m m^3-s. In the continuity equation (Equation 1–1 in the Theory Guide), the defined mass source will appear in the S_m term.

If you have more than one species, you can specify mass sources for each individual species. There will be a total **Mass** source term as well as a source term listed explicitly for each species (e.g., **h2**, **o2**) except the last one you defined. If the total of all species mass sources (including the last one) is 0, then you should specify a value of 0 for the **Mass** source, and also specify the values of the non-zero individual species mass sources. Since you cannot specify the mass source for the last species explicitly, ANSYS FLUENT will compute it by subtracting the sum of all other species mass sources from the specified total **Mass** source.

For example, if the mass source for hydrogen in a hydrogen-air mixture is 0.01, the mass source for oxygen is 0.02, and the mass source for nitrogen (the last species) is 0.015, you will specify a value of 0.01 in the **h2** field, a value of 0.02 in the **o2** field, and a value of 0.045 in the **Mass** field. This concept also applies within each cell if you use user-defined functions for species mass sources.

The units for the species mass sources are kg/m m^3-s. In the conservation equation for a chemical species (Equation 7–1 in the Theory Guide), the defined mass source will appear in the S_i term.

Mass sources can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

7.2.5.2.2. Momentum Sources

To define a source of momentum, specify the **X Momentum**, **Y Momentum**, and/or **Z Momentum** term. The units for the momentum source are N/m^3. In the momentum equation (Equation 1–3 in the Theory Guide), the defined momentum source will appear in the $\vec{F}$ term.

Momentum sources can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.5.2.3. Energy Sources

To define a source of energy, specify an **Energy** term. The units for the energy source are W/m^3. In the energy equation (Equation 5–1 in the Theory Guide), the defined energy source will appear in the S_h term.

Energy sources can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

7.2.5.2.4. Turbulence Sources

7.2.5.2.5. Turbulence Sources for the k- ε Model

To define a source of k or ε in the k- ε equations, specify the **Turbulent Kinetic Energy** or **Turbulent Dissipation Rate** term. The units for the k source are kg/m-s^3 and those for ε are kg/m-s^4.

The defined k source will appear in the S_k term on the right-hand side of the turbulent kinetic energy equation (e.g., Equation 4–33 in the Theory Guide).

The defined ε source will appear in the S_ε term on the right-hand side of the turbulent dissipation rate equation (e.g., Equation 4–34 in the Theory Guide).

7.2.5.2.6. Turbulence Sources for the Spalart-Allmaras Model

To define a source of modified turbulent viscosity, specify the **Modified Turbulent Viscosity** term. The units for the modified turbulent viscosity source are kg/m-s^2. In the transport equation for the Spalart-Allmaras model (Equation 4–15 in the Theory Guide), the defined modified turbulent viscosity source will appear in the $S_{\tilde{\nu}}$ term.

7.2.5.2.7. Turbulence Sources for the k- ω Model

To define a source of k or ω in the k- ω equations, specify the **Turbulent Kinetic Energy** or **Specific Dissipation Rate** term. The units for the k source are kg/m-s^3 and those for ω are kg/m^3-s^2.

The defined k source will appear in the S_k term on the right-hand side of the turbulent kinetic energy equation (Equation 4–64 in the Theory Guide).

The defined ω source will appear in the S_ω term on the right-hand side of the specific turbulent dissipation rate equation (Equation 4–65 in the Theory Guide).

7.2.5.2.8. Turbulence Sources for the Reynolds Stress Model

To define a source of k, ε, or the Reynolds stresses in the RSM transport equations, specify the **Turbulence Kinetic Energy**, **Turbulence Dissipation Rate**, **UU Reynolds Stress**, **VV Reynolds Stress**, **WW Reynolds Stress**, **UV Reynolds Stress**, **VW Reynolds Stress**, and/or **UW Reynolds Stress** terms. The units for the k source and the sources of Reynolds stress are kg/m-s^3, and those for ε are kg/m-s^4.

The defined Reynolds stress sources will appear in the S_{user} term on the right-hand side of the Reynolds stress transport equation (Equation 4–185 in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The defined k source will appear in the S_k term on the right-hand side of Equation 4–212 in the Theory Guide.

The defined ε will appear in the S_ε term on the right-hand side of Equation 4–215 in the Theory Guide.

7.2.5.2.9. Mean Mixture Fraction and Variance Sources

To define a source of the mean mixture fraction or its variance for the non-premixed combustion model, specify the **Mean Mixture Fraction** or **Mixture Fraction Variance** term. The units for the mean mixture fraction source are kg/mm^3-s, and those for the mixture fraction variance source are kg/m^3-s.

The defined mean mixture fraction source will appear in the S_{user} term in the transport equation for the mixture fraction (Equation 8–4 in the Theory Guide).

The defined mixture fraction variance source will appear in the S_{user} term in the transport equation for the mixture fraction variance (Equation 8–5 in the Theory Guide).

If you are using the two-mixture-fraction approach, you can also specify sources of the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**.

7.2.5.2.10. P-1 Radiation Sources

To define a source for the P-1 radiation model, specify the **P1** term. The units for the radiation source are W/m^3, and the defined source will appear in the S_G term in Equation 5–18 in the Theory Guide.

Note that, if the source term you are defining represents a transfer from internal energy to radiative energy (e.g., absorption or emission), you will need to specify an **Energy** source of the same magnitude as the **P1** source, but with the opposite sign, in order to ensure overall energy conservation.

7.2.5.2.11. Progress Variable Sources

To define a source of the progress variable for the premixed combustion model, specify the **Progress Variable** term. The units for the progress variable source are kg/m^3-s, and the defined source will appear in the ρS_c term in Equation 9–1 in the Theory Guide.

7.2.5.2.12. NO, HCN, and NH3 Sources for the NOx Model

To define a source of NO, HCN, or NH_3 for the NOx model, specify the **no**, **hcn**, or **nh3** term. The units for these sources are kg/m^3-s, and the defined sources will appear in the S_{NO}, S_{HCN}, and S_{NH3} terms of Equation 14–1, Equation 14–2, and Equation 14–3 in the Theory Guide.

7.2.5.2.13. User-Defined Scalar (UDS) Sources

You can specify source term(s) for each UDS transport equation you have defined in your model. See *Setting Up UDS Equations in ANSYS FLUENT* (p. 488) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3. Boundary Conditions

Boundary conditions consist of flow inlets and exit boundaries, wall, repeating, and pole boundaries, and internal face boundaries. All the various types of boundary conditions are discussed in the sections that follow.

7.3.1. Flow Inlet and Exit Boundary Conditions
7.3.2. Using Flow Boundary Conditions
7.3.3. Pressure Inlet Boundary Conditions
7.3.4. Velocity Inlet Boundary Conditions
7.3.5. Mass Flow Inlet Boundary Conditions
7.3.6. Inlet Vent Boundary Conditions
7.3.7. Intake Fan Boundary Conditions
7.3.8. Pressure Outlet Boundary Conditions
7.3.9. Pressure Far-Field Boundary Conditions
7.3.10. Outflow Boundary Conditions
7.3.11. Outlet Vent Boundary Conditions
7.3.12. Exhaust Fan Boundary Conditions
7.3.13. Wall Boundary Conditions
7.3.14. Symmetry Boundary Conditions
7.3.15. Periodic Boundary Conditions
7.3.16. Axis Boundary Conditions
7.3.17. Fan Boundary Conditions
7.3.18. Radiator Boundary Conditions
7.3.19. Porous Jump Boundary Conditions

7.3.1. Flow Inlet and Exit Boundary Conditions

ANSYS FLUENT has a wide range of boundary conditions that permit flow to enter and exit the solution domain. To help you select the most appropriate boundary condition for your application, this section includes descriptions of how each type of condition is used, and what information is needed for each one. Recommendations for determining inlet values of the turbulence parameters are also provided.

7.3.2. Using Flow Boundary Conditions

This section provides an overview of flow boundaries in ANSYS FLUENT and how to use them.

ANSYS FLUENT provides 10 types of boundary zone types for the specification of flow inlets and exits: velocity inlet, pressure inlet, mass flow inlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, and exhaust fan.

The inlet and exit boundary condition options in ANSYS FLUENT are as follows:

- Velocity inlet boundary conditions are used to define the velocity and scalar properties of the flow at inlet boundaries.
- Pressure inlet boundary conditions are used to define the total pressure and other scalar quantities at flow inlets.
- Mass flow inlet boundary conditions are used in compressible flows to prescribe a mass flow rate at an inlet. It is not necessary to use mass flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow. Like pressure and velocity inlets, other inlet scalars are also prescribed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Pressure outlet boundary conditions are used to define the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a better rate of convergence when backflow occurs during iteration.
- Pressure far-field boundary conditions are used to model a free-stream compressible flow at infinity, with free-stream Mach number and static conditions specified. This boundary type is available only for compressible flows.
- Outflow boundary conditions are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. They are appropriate where the exit flow is close to a fully developed condition, as the outflow boundary condition assumes a zero streamwise gradient for all flow variables except pressure. They are not appropriate for compressible flow calculations.
- Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) total pressure and temperature.
- Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) total pressure and temperature.
- Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) static pressure and temperature.
- Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) static pressure.

7.3.2.1. Determining Turbulence Parameters

When the flow enters the domain at an inlet, outlet, or far-field boundary, ANSYS FLUENT requires specification of transported turbulence quantities. This section describes which quantities are needed for specific turbulence models and how they must be specified. It also provides guidelines for the most appropriate way of determining the inflow boundary values.

7.3.2.1.1. Specification of Turbulence Quantities Using Profiles

If it is important to accurately represent a boundary layer or fully-developed turbulent flow at the inlet, you should ideally set the turbulence quantities by creating a profile file (see *Profiles* (p. 366)) from experimental data or empirical formulas. If you have an analytical description of the profile, rather than data points, you can either use this analytical description to create a profile file, or create a user-defined function to provide the inlet boundary information. (See the UDF Manual for information on user-defined functions.)

Once you have created the profile function, you can use it as described below:

- Spalart-Allmaras model: Choose **Turbulent Viscosity** or **Turbulent Viscosity Ratio** in the **Turbulence Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to **Turbulent Viscosity** or **Turbulent Viscosity Ratio**. ANSYS FLUENT computes the boundary value for the modified turbulent viscosity, $\tilde{\nu}$, by combining μ_t/μ with the appropriate values of density and molecular viscosity.
- k- ε models: Choose **K and Epsilon** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**.
- k- ω models: Choose **K and Omega** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Specific Dissipation Rate**.
- Reynolds stress model: Choose **K and Epsilon** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **Turbulent Dissipation Rate**. Choose **Reynolds-Stress Components** in the **Reynolds-Stress Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to each of the individual Reynolds-stress components.

7.3.2.1.2. Uniform Specification of Turbulence Quantities

In some situations, it is appropriate to specify a uniform value of the turbulence quantity at the boundary where inflow occurs. Examples are fluid entering a duct, far-field boundaries, or even fully-developed duct flows where accurate profiles of turbulence quantities are unknown.

In most turbulent flows, higher levels of turbulence are generated within shear layers than enter the domain at flow boundaries, making the result of the calculation relatively insensitive to the inflow boundary values. Nevertheless, caution must be used to ensure that boundary values are not so unphysical as to contaminate your solution or impede convergence. This is particularly true of external flows where unphysically large values of effective viscosity in the free stream can "swamp" the boundary layers.

You can use the turbulence specification methods described above to enter uniform constant values instead of profiles. Alternatively, you can specify the turbulence quantities in terms of more convenient quantities such as turbulence intensity, turbulent viscosity ratio, hydraulic diameter, and turbulence length scale. These quantities are discussed further in the following sections.

7.3.2.1.3. Turbulence Intensity

The turbulence intensity, I, is defined as the ratio of the root-mean-square of the velocity fluctuations, u', to the mean flow velocity, u_{avg}.

A turbulence intensity of 1% or less is generally considered low and turbulence intensities greater than 10% are considered high. Ideally, you will have a good estimate of the turbulence intensity at the inlet boundary from external, measured data. For example, if you are simulating a wind-tunnel experiment, the turbulence intensity in the free stream is usually available from the tunnel characteristics. In modern low-turbulence wind tunnels, the free-stream turbulence intensity may be as low as 0.05%.

For internal flows, the turbulence intensity at the inlets is totally dependent on the upstream history of the flow. If the flow upstream is under-developed and undisturbed, you can use a low turbulence intensity. If the flow is fully developed, the turbulence intensity may be as high as a few percent. The turbulence intensity at the core of a fully-developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows:

$$I \equiv \frac{u'}{u_{avg}} = 0.16\left(Re_{D_H}\right)^{-1/8} \qquad \textbf{(7–41)}$$

At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

7.3.2.1.4. Turbulence Length Scale and Hydraulic Diameter

The turbulence length scale, ℓ, is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows.

In fully-developed duct flows, ℓ is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between ℓ and the physical size of the duct is

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\ell = 0.07L \tag{7–42}$$

where L is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where L is the diameter of the pipe. In a channel of non-circular cross-section, you can base L on the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on the duct size.

It should be noted that the relationship of *Equation 7–42* (p. 252), which relates a physical dimension (L) to the turbulence length scale (ℓ), is not necessarily applicable to all situations. For most cases, however, it is a suitable approximation.

Guidelines for choosing the characteristic length L or the turbulence length scale ℓ for selected flow types are listed below:

- For fully-developed internal flows, choose the **Intensity and Hydraulic Diameter** specification method and specify the hydraulic diameter $L = D_H$ in the **Hydraulic Diameter** field.
- For flows downstream of turning vanes, perforated plates, etc., choose the **Intensity and Length Scale** method and specify the characteristic length of the flow opening for L in the **Turbulent Length Scale** field.
- For wall-bounded flows in which the inlets involve a turbulent boundary layer, choose the **Intensity and Length Scale** method and use the boundary-layer thickness, δ_{99}, to compute the turbulence length scale, ℓ, from $\ell = 0.4\delta_{99}$. Enter this value for ℓ in the **Turbulence Length Scale** field.

7.3.2.1.5. Turbulent Viscosity Ratio

The turbulent viscosity ratio, μ_t/μ, is directly proportional to the turbulent Reynolds number ($Re_t \equiv k^2/(\varepsilon\nu)$). Re_t is large (on the order of 100 to 1000) in high-Reynolds-number boundary layers, shear layers, and fully-developed duct flows. However, at the free-stream boundaries of most external flows, μ_t/μ is fairly small. Typically, the turbulence parameters are set so that $1 < \mu_t/\mu < 10$.

To specify quantities in terms of the turbulent viscosity ratio, you can choose **Turbulent Viscosity Ratio** (for the Spalart-Allmaras model) or **Intensity and Viscosity Ratio** (for the k-ε models, the k-ω models, or the RSM).

7.3.2.1.6. Relationships for Deriving Turbulence Quantities

To obtain the values of transported turbulence quantities from more convenient quantities such as I, L, or μ_t/μ, you must typically resort to an empirical relation. Several useful relations, most of which are used within ANSYS FLUENT, are presented below.

7.3.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale

To obtain the modified turbulent viscosity, $\tilde{\nu}$, for the Spalart-Allmaras model from the turbulence intensity, I, and length scale, ℓ, the following equation can be used:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tilde{\nu} = \sqrt{\frac{3}{2}} u_{avg} I \ell \tag{7–43}$$

This formula is used in ANSYS FLUENT if you select the **Intensity and Hydraulic Diameter** specification method with the Spalart-Allmaras model. ℓ is obtained from *Equation 7–42* (p. 252).

7.3.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity

The relationship between the turbulent kinetic energy, k, and turbulence intensity, I, is

$$k = \frac{3}{2}\left(u_{avg} I\right)^2 \tag{7–44}$$

where u_{avg} is the mean flow velocity.

This relationship is used in ANSYS FLUENT whenever the **Intensity and Hydraulic Diameter**, **Intensity and Length Scale**, or **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ε.

7.3.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ, you can determine ε from the relationship

$$\varepsilon = C_\mu^{3/4} \frac{k^{3/2}}{\ell} \tag{7–45}$$

where C_μ is an empirical constant specified in the turbulence model (approximately 0.09). The determination of ℓ was discussed previously.

This relationship is used in ANSYS FLUENT whenever the **Intensity and Hydraulic Diameter** or **Intensity and Length Scale** method is used instead of specifying explicit values for k and ε.

7.3.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio

The value of ε can be obtained from the turbulent viscosity ratio μ_t/μ and k using the following relationship:

$$\varepsilon = \rho C_\mu \frac{k^2}{\mu}\left(\frac{\mu_t}{\mu}\right)^{-1} \tag{7–46}$$

where C_μ is an empirical constant specified in the turbulence model (approximately 0.09).

This relationship is used in ANSYS FLUENT whenever the **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ε.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence

If you are simulating a wind-tunnel situation in which the model is mounted in the test section downstream of a mesh and/or wire mesh screens, you can choose a value of ε such that

$$\varepsilon \approx \frac{\Delta k U_\infty}{L_\infty} \tag{7–47}$$

where Δk is the approximate decay of k you wish to have across the flow domain (say, 10% of the inlet value of k), U_∞ is the free-stream velocity, and L_∞ is the streamwise length of the flow domain. *Equation 7–47* (p. 254) is a linear approximation to the power-law decay observed in high-Reynolds-number isotropic turbulence. Its basis is the exact equation for k in decaying turbulence, $U \partial k / \partial x = -\varepsilon$.

If you use this method to estimate ε, you should also check the resulting turbulent viscosity ratio μ_t / μ to make sure that it is not too large, using *Equation 7–46* (p. 253).

Although this method is not used internally by ANSYS FLUENT, you can use it to derive a constant free-stream value of ε that you can then specify directly by choosing **K and Epsilon** in the **Turbulence Specification Method** drop-down list. In this situation, you will typically determine k from I using *Equation 7–44* (p. 253).

7.3.2.1.12. Estimating Specific Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ, you can determine ω from the relationship

$$\omega = \frac{k^{1/2}}{C_\mu^{1/4} \ell} \tag{7–48}$$

where C_μ is an empirical constant specified in the turbulence model (approximately 0.09). The determination of ℓ was discussed previously.

This relationship is used in ANSYS FLUENT whenever the **Intensity and Hydraulic Diameter** or **Intensity and Length Scale** method is used instead of specifying explicit values for k and ω.

7.3.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio

The value of ω can be obtained from the turbulent viscosity ratio μ_t / μ and k using the following relationship:

$$\omega = \rho \frac{k}{\mu} \left(\frac{\mu_t}{\mu} \right)^{-1} \tag{7–49}$$

This relationship is used in ANSYS FLUENT whenever the **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ω.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy

When the RSM is used, if you do not specify the values of the Reynolds stresses explicitly at the inlet using the **Reynolds-Stress Components** option in the **Reynolds-Stress Specification Method** drop-down list, they are approximately determined from the specified values of k. The turbulence is assumed to be isotropic such that

$$\overline{u'_i u'_j} = 0 \tag{7–50}$$

and

$$\overline{u'_\alpha u'_\alpha} = \frac{2}{3}k \tag{7–51}$$

(no summation over the index α).

ANSYS FLUENT will use this method if you select **K or Turbulence Intensity** in the **Reynolds-Stress Specification Method** drop-down list.

7.3.2.1.15. Specifying Inlet Turbulence for LES

The turbulence intensity value specified at a velocity inlet for LES, as described in *Large Eddy Simulation Model* (p. 680), is used to randomly perturb the instantaneous velocity field at the inlet. It does not specify a modeled turbulence quantity. Instead, the stochastic components of the flow at the inlet boundary are accounted for by superposing random perturbations on individual velocity components as described in Inlet Boundary Conditions for the LES Model in the Theory Guide.

7.3.3. Pressure Inlet Boundary Conditions

Pressure inlet boundary conditions are used to define the fluid pressure at flow inlets, along with all other scalar properties of the flow. They are suitable for both incompressible and compressible flow calculations. Pressure inlet boundary conditions can be used when the inlet pressure is known but the flow rate and/or velocity is not known. This situation may arise in many practical situations, including buoyancy-driven flows. Pressure inlet boundary conditions can also be used to define a “free” boundary in an external or unconfined flow.

For an overview of flow boundaries, see *Flow Inlet and Exit Boundary Conditions* (p. 249).

7.3.3.1. Inputs at Pressure Inlet Boundaries

7.3.3.1.1. Summary

You will enter the following information for a pressure inlet boundary:

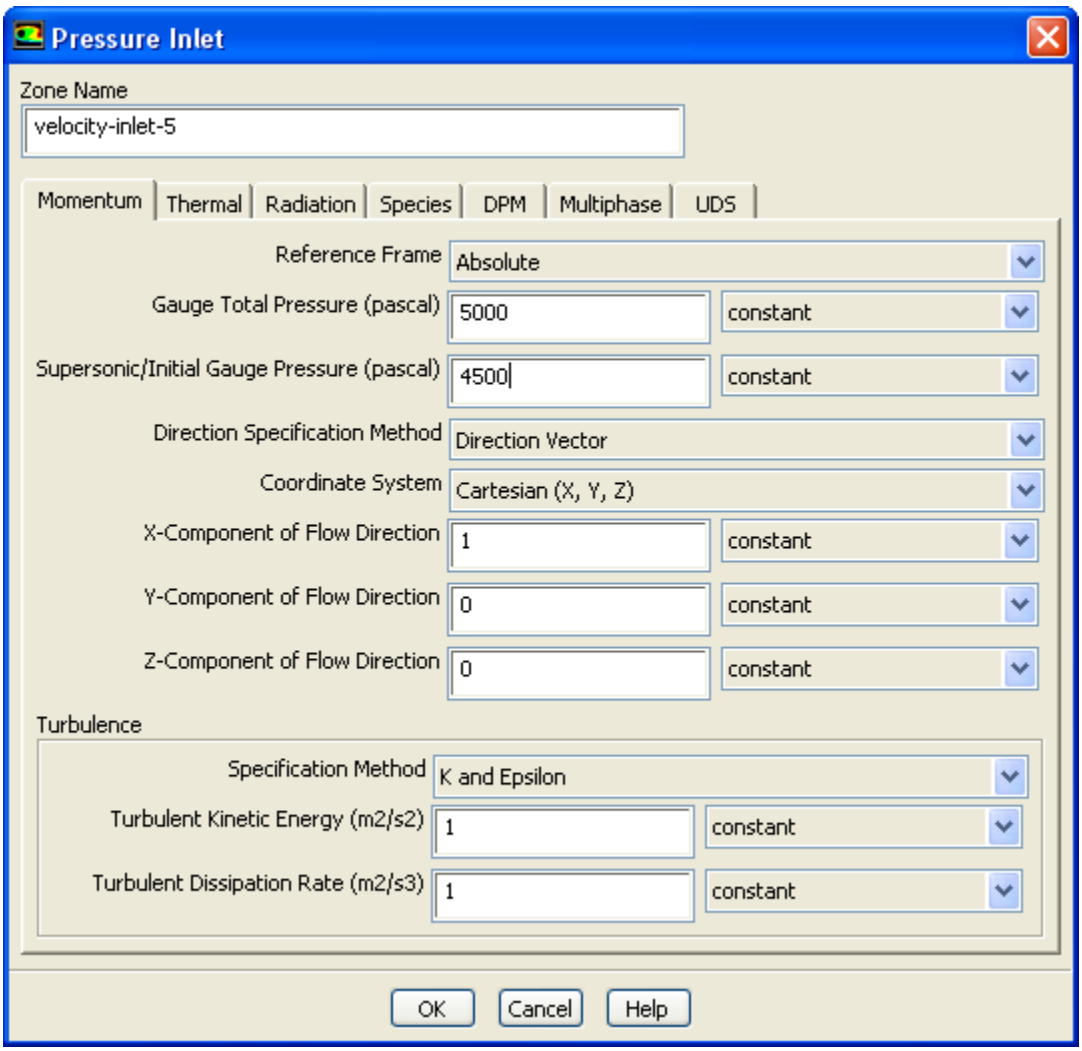
- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the *Pressure Inlet Dialog Box* (p. 1848) (*Figure 7.12* (p. 256)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)). Note that open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

Figure 7.12 The Pressure Inlet Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.3.1.1.1. Pressure Inputs and Hydrostatic Head

When gravitational acceleration is activated in the **Operating Conditions** dialog box (accessed from the **Boundary Conditions** task page), the pressure field (including all pressure inputs) will include the hydrostatic head. This is accomplished by redefining the pressure in terms of a modified pressure which includes the hydrostatic head (denoted p') as follows:

$$p' = p + \rho_0 \vec{g} \cdot \vec{r} \tag{7–52}$$

where ρ_0 is a constant reference density, $\vec{g}$ is the gravity vector (also a constant), and

$$\vec{r} = x\hat{\imath} + y\hat{\jmath} + z\hat{k} \tag{7–53}$$

is the position vector. Noting that

$$\nabla \left(\rho_0 \vec{g} \cdot \vec{r} \right) = \rho_0 \vec{g} \tag{7–54}$$

it follows that

$$\nabla p' = \nabla \left(p + \rho_0 \vec{g} \cdot \vec{r} \right) = \nabla p - \rho_0 \vec{g} \tag{7–55}$$

The substitution of this relation in the momentum equation gives pressure gradient and gravitational body force terms of the form

$$\nabla p' + \left(\rho - \rho_0 \right) \vec{g} \tag{7–56}$$

where ρ is the fluid density. Therefore, if the fluid density is constant, we can set the reference density ρ_0 equal to the fluid density, thereby eliminating the body force term. If the fluid density is not constant (for example, density is given by the ideal gas law), then the reference density should be chosen to be representative of the average or mean density in the fluid domain, so that the body force term is small.

An important consequence of this treatment of the gravitational body force is that your inputs of pressure (now defined as p') should not include hydrostatic pressure differences. Moreover, reports of static and total pressure will not show any influence of the hydrostatic pressure. See *Natural Convection and Buoyancy-Driven Flows* (p. 697) for additional information.

7.3.3.1.1.2. Defining Total Pressure and Temperature

Enter the value for total pressure in the **Gauge Total Pressure** field in the **Pressure Inlet** dialog box. Total temperature is set in the **Thermal** tab, in the **Total Temperature** field.

Remember that the total pressure value is the gauge pressure with respect to the operating pressure defined in the *Operating Conditions Dialog Box* (p. 1806). Total pressure for an incompressible fluid is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$p_0 = p_s + \frac{1}{2}\rho |\vec{v}|^2 \tag{7–57}$$

and for a compressible fluid of constant c_p as

$$p_0 = p_s \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/(\gamma-1)} \tag{7–58}$$

where p_0 = total pressure

p_s = static pressure

M = Mach number

γ = ratio of specific heats $\left(c_p/c_v\right)$

If you are modeling axisymmetric swirl, $\vec{v}$ in *Equation 7–57* (p. 258) will include the swirl component.

If the cell zone adjacent to a pressure inlet is defined as a moving reference frame zone, and you are using the pressure-based solver, the velocity in *Equation 7–57* (p. 258) (or the Mach number in *Equation 7–58* (p. 258)) will be absolute or relative to the mesh velocity, depending on whether or not the **Absolute** velocity formulation is enabled in the **General** task page. For the density-based solver, the **Absolute** velocity formulation is always used; hence, the velocity in *Equation 7–57* (p. 258) (or the Mach number in *Equation 7–58* (p. 258)) is always the **Absolute** velocity.

- If **Reference Frame** is set to **Absolute** in the **Pressure Inlet** dialog box, then the total temperature, total pressure, and flow direction are also in the absolute reference frame, and therefore, the ANSYS FLUENT solver will convert it to the relative reference frame.
- If **Reference Frame** is set to **Relative to Adjacent Cell Zone** in the **Pressure Inlet** dialog box, then the total temperature, total pressure, and velocity components are also relative to the adjacent cell zone and no change is needed.

For the Eulerian multiphase model, the total temperature, and velocity components need to be specified for the individual phases. The **Reference Frame** (**Relative to Adjacent Cell Zone** or **Absolute**) for each of the phases is the same as the reference frame selected for the mixture phase. Note that the total pressure values need to be specified in the mixture phase.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- If the flow is incompressible, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the static temperature.
- For the mixture multiphase model, if a boundary allows a combination of compressible and incompressible phases to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the static temperature at that boundary. If a boundary allows *only a compressible* phase to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be taken as the total temperature (relative/absolute) at that boundary. The total temperature will depend on the **Reference Frame** option selected in the **Pressure Inlet** dialog box.
- For the VOF multiphase model, if a boundary allows a *compressible phase* to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the total temperature at that boundary. The total temperature (relative/absolute) will depend on the **Reference Frame** option chosen in the dialog box. Otherwise, the temperature assigned to the boundary will be considered the static temperature at the boundary.
- For the Eulerian multiphase model, if a boundary allows a mixture of compressible and incompressible phases in the domain, then the temperature of each of the phases will be the total or static temperature, depending on whether the phase is compressible or incompressible.
- Total temperature (relative/absolute) will depend on the **Reference Frame** option chosen in the **Pressure Inlet** dialog box.

7.3.3.1.1.3. Defining the Flow Direction

The flow direction is defined as a unit vector ($\vec{d}$) which is aligned with the local velocity vector, $\vec{v}$. This can be expressed simply as

$$\vec{d} = \frac{\vec{v}}{|\vec{v}|} \tag{7–59}$$

Important

For the inputs in ANSYS FLUENT, the flow direction $\vec{d}$ need not be a unit vector, as it will be automatically normalized before it is applied.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

For a moving reference frame, the relative flow direction $\vec{d}_r$ is defined in terms of the relative velocity, $\vec{v}_r$. Thus,

$$\vec{d}_r = \frac{\vec{v}_r}{|\vec{v}_r|} \tag{7–60}$$

You can define the flow direction at a pressure inlet explicitly, or you can define the flow to be normal to the boundary. If you choose to specify the direction vector, you can set either the (Cartesian) x, y, and $z$components, or the (cylindrical) radial, tangential, and axial components.

For moving zone problems calculated using the pressure-based solver, the flow direction will be absolute or relative to the mesh velocity, depending on whether or not the **Absolute** velocity formulation is enabled in the **General** task page. For the density-based solver, the flow direction will always be in the absolute frame.

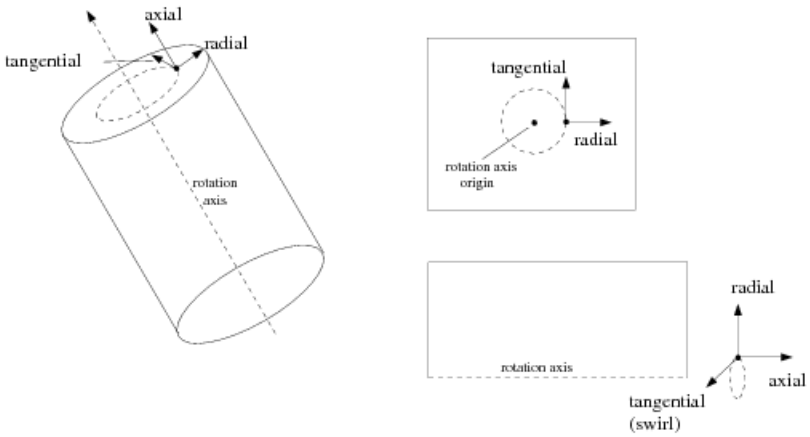
The procedure for defining the flow direction is as follows (refer to *Figure 7.12* (p. 256)):

1. Specify the flow direction by selecting **Direction Vector** or **Normal to Boundary** in the **Direction Specification Method** drop-down list.
2. If you selected **Normal to Boundary** in step 1 and you are modeling axisymmetric swirl, enter the appropriate value for the **Tangential-Component of Flow Direction**. If you chose **Normal to Boundary** and your geometry is 3D or 2D without axisymmetric swirl, there are no additional inputs for flow direction.
3. If you selected **Direction Vector** in step 1, and your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical(Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** from the **Coordinate System** drop-down list. Some notes on these selections are provided below:
   - The **Cartesian** coordinate option is based on the Cartesian coordinate system used by the geometry. Enter appropriate values for the **X**, **Y**, and **Z-Component of Flow Direction**.
   - The **Cylindrical** coordinate system uses the axial, radial, and tangential components based on the following coordinate systems:
     - For problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the *Fluid Dialog Box* (p. 1796).
     - For problems involving multiple zones (e.g., multiple reference frames or sliding meshes), the coordinate system is defined by the rotation axis specified in the **Fluid** (or **Solid**) dialog box for the fluid (or solid) zone that is adjacent to the inlet.

     For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially outward from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis (see *Figure 7.13* (p. 261)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.13 Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains



- The **Local Cylindrical** coordinate system allows you to define a coordinate system specifically for the inlet. When you use the local cylindrical option, you will define the coordinate system right here in the **Pressure Inlet** dialog box. The local cylindrical coordinate system is useful if you have several inlets with different rotation axes. Enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of the Axis Origin and Axis Direction.
- The **Local Cylindrical Swirl** coordinate system option allows you to define a coordinate system specifically for the inlet where the total pressure, swirl velocity, and the components of the velocity in the axial and radial planes are specified. Enter appropriate values for the **Axial** and **Radial-Component of Flow Direction**, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the Axis Origin and Axis Direction. It is recommended that you start your simulation with a smaller swirl velocity and then progressively increase the velocity to obtain a stable solution.

Important

Local Cylindrical Swirl should not be used for open channel boundary conditions and on the mixing plane boundaries while using the mixing plane model.

4. If you selected **Direction Vector** in step 1, and your geometry is 2D, define the vector components as follows:
   - For a 2D planar geometry, enter appropriate values for the **X**, **Y**, and **Z-Component of Flow Direction**.
   - For a 2D axisymmetric geometry, enter appropriate values for the **Axial**, **Radial-Component of Flow Direction**.
   - For a 2D axisymmetric swirl geometry, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**.

   Figure 7.13 (p. 261) shows the vector components for these different coordinate systems.

7.3.3.1.1.4. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in *Initializing the Solution* (p. 1239).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remember that the static pressure value you enter is relative to the operating pressure set in the *Operating Conditions Dialog Box* (p. 1806). Note the comments in *Pressure Inputs and Hydrostatic Head* (p. 257) regarding hydrostatic pressure.

The **Supersonic/Initial Gauge Pressure** is ignored by ANSYS FLUENT whenever the flow is subsonic, in which case it is calculated from the specified stagnation quantities. If you choose to initialize the solution based on the pressure-inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation pressure to compute initial values according to the isentropic relations (for compressible flow) or Bernoulli's equation (for incompressible flow). Therefore, for a sub-sonic inlet it should generally be set based on a reasonable estimate of the inlet Mach number (for compressible flow) or inlet velocity (for incompressible flow).

7.3.3.1.1.5. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in *Determining Turbulence Parameters* (p. 250). Turbulence modeling in general is described in *Modeling Turbulence* (p. 643).

7.3.3.1.1.6. Defining Radiation Parameters

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See *Defining Boundary Conditions for Radiation* (p. 724) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

7.3.3.1.1.7. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see *Defining Cell Zone and Boundary Conditions for Species* (p. 822).

7.3.3.1.1.8. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in *Defining Non-Premixed Boundary Conditions* (p. 883).

7.3.3.1.1.9. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in *Setting Boundary Conditions for the Progress Variable* (p. 899).

7.3.3.1.1.10. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure inlet. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.3.1.1.11. Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.3.1.1.12. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See *Modeling Open Channel Flows* (p. 1121) for details.

7.3.3.2. Default Settings at Pressure Inlet Boundaries

Default settings (in SI) for pressure inlet boundary conditions are as follows:

Gauge Total Pressure	0
Supersonic/Initial Gauge Pressure	0
Total Temperature	300
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
Turbulent Kinetic Energy	1
Turbulent Dissipation Rate	1

7.3.3.3. Calculation Procedure at Pressure Inlet Boundaries

The treatment of pressure inlet boundary conditions by ANSYS FLUENT can be described as a loss-free transition from stagnation conditions to the inlet conditions. For incompressible flows, this is accomplished by application of the Bernoulli equation at the inlet boundary. In compressible flows, the equivalent isentropic flow relations for an ideal gas are used.

7.3.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries

When flow enters through a pressure inlet boundary, ANSYS FLUENT uses the boundary condition pressure you input as the total pressure of the fluid at the inlet plane, p_0. In incompressible flow, the inlet total pressure and the static pressure, p_s, are related to the inlet velocity via Bernoulli's equation:

$$p_0 = p_s + \frac{1}{2}\rho v^2 \qquad \textbf{(7–61)}$$

With the resulting velocity magnitude and the flow direction vector you assigned at the inlet, the velocity components can be computed. The inlet mass flow rate and fluxes of momentum, energy, and species can then be computed as outlined in *Calculation Procedure at Velocity Inlet Boundaries* (p. 269).

For incompressible flows, density at the inlet plane is either constant or calculated as a function of temperature and/or species mass/mole fractions, where the mass or mole fractions are the values you entered as an inlet condition.

If flow exits through a pressure inlet, the total pressure specified is used as the static pressure. For incompressible flows, total temperature is equal to static temperature.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries

In compressible flows, isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, p_0', at the inlet and the static pressure, p_s', in the adjacent fluid cell are thus related as

$$\frac{p_0'+p_{op}}{p_s'+p_{op}}=\left(1+\frac{\gamma-1}{2}M^2\right)^{\gamma/(\gamma-1)} \tag{7–62}$$

where

$$M\equiv\frac{v}{c}=\frac{v}{\sqrt{\gamma R T_s}} \tag{7–63}$$

c = the speed of sound, and $\gamma=c_p/c_v$. Note that the operating pressure, p_{op}, appears in *Equation 7–62* (p. 264) because your boundary condition inputs are in terms of pressure relative to the operating pressure. Given p_0' and p_s', *Equation 7–62* (p. 264) and *Equation 7–63* (p. 264) are used to compute the velocity magnitude of the fluid at the inlet plane. Individual velocity components at the inlet are then derived using the direction vector components.

For compressible flow, the density at the inlet plane is defined by the ideal gas law in the form

$$\rho=\frac{p_s'+p_{op}}{RT_s} \tag{7–64}$$

For multi-species gas mixtures, the specific gas constant, R, is computed from the species mass or mole fractions, Yi that you defined as boundary conditions at the pressure inlet boundary. The static temperature at the inlet, T_s, is computed from your input of total temperature, T_0, as

$$\frac{T_0}{T_s}=1+\frac{\gamma-1}{2}M^2 \tag{7–65}$$

7.3.4. Velocity Inlet Boundary Conditions

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. In this case, the total (or stagnation) pressure is not fixed but will rise (in response to the computed static pressure) to whatever value is necessary to provide the prescribed velocity distribution. This boundary condition is equally applicable to incompressible and compressible flows.

In special instances, a velocity inlet may be used in ANSYS FLUENT to define the flow velocity at flow exits. (The scalar inputs are not used in such cases.) In such cases you must ensure that overall continuity is maintained in the domain.

For an overview of flow boundaries, see *Flow Inlet and Exit Boundary Conditions* (p. 249).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.4.1. Inputs at Velocity Inlet Boundaries

7.3.4.1.1. Summary

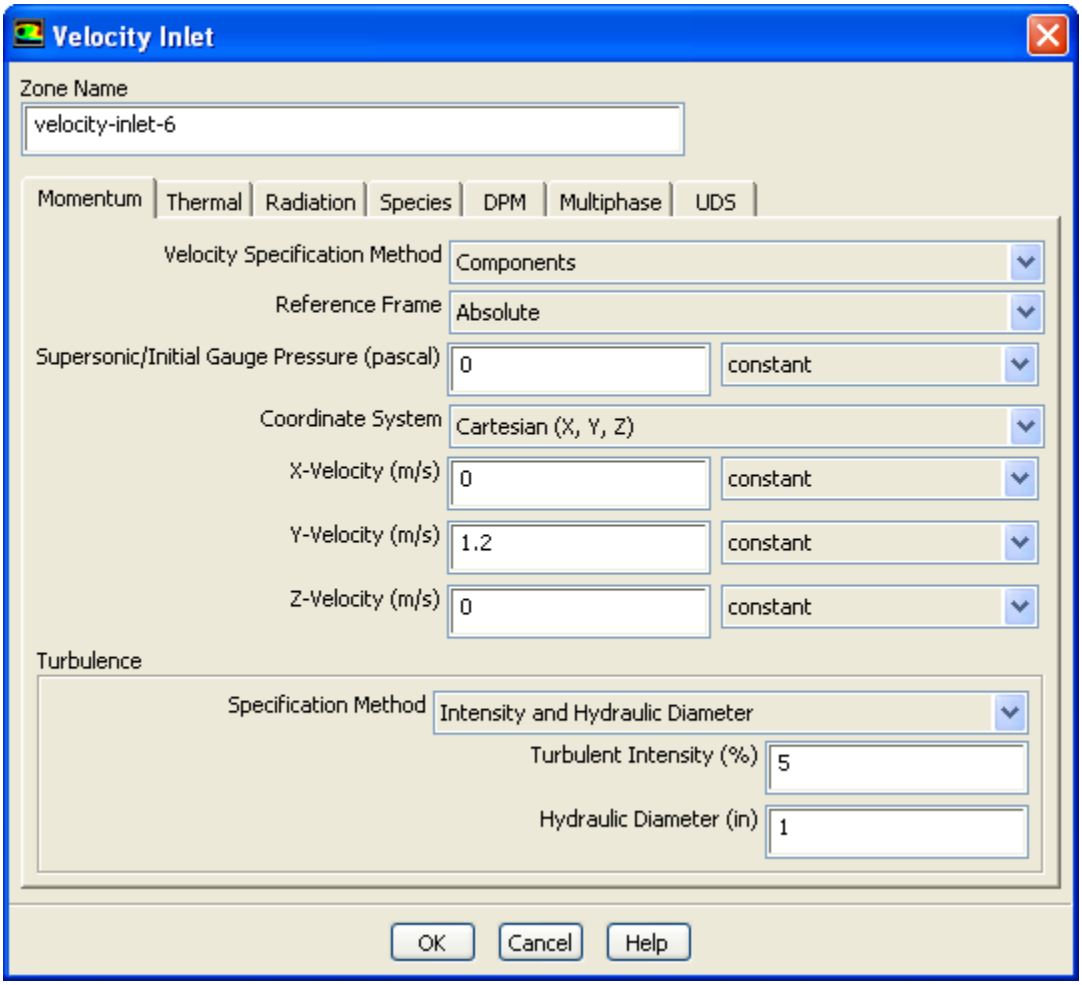
You will enter the following information for a velocity inlet boundary:

- type of reference frame
- velocity magnitude and direction or velocity components
- swirl velocity (for 2D axisymmetric problems with swirl)
- static pressure
- temperature (for energy calculations)
- outflow gauge pressure (for calculations with the density-based solver)
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)

All values are entered in the *Velocity Inlet Dialog Box* (p. 1860) (*Figure 7.14* (p. 266)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.14 The Velocity Inlet Dialog Box



7.3.4.1.2. Defining the Velocity

You can define the inflow velocity by specifying the velocity magnitude and direction, the velocity components, or the velocity magnitude normal to the boundary. If the cell zone adjacent to the velocity inlet is moving (i.e., if you are using a moving reference frame, multiple reference frames, or sliding meshes), you can specify either relative or absolute velocities. For axisymmetric problems with swirl in ANSYS FLUENT, you will also specify the swirl velocity.

The procedure for defining the inflow velocity is as follows:

1. Specify the flow direction by selecting **Magnitude and Direction**, **Components**, or **Magnitude, Normal to Boundary** in the **Velocity Specification Method** drop-down list.
2. If the cell zone adjacent to the velocity inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list. If the adjacent cell zone is not moving, **Absolute** and **Relative to Adjacent Cell Zone** will be equivalent, so you need not visit the list.
3. If you are going to set the velocity magnitude and direction or the velocity components, and your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical (Radial, Tangential, Axial)** from the **Coordinate System** drop-down list. See *Defining the Flow Direction* (p. 259) for information about Cartesian, cylindrical, and local cylindrical coordinate systems.
4. Set the appropriate velocity parameters, as described below for each specification method.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.4.1.3. Setting the Velocity Magnitude and Direction

If you selected **Magnitude and Direction** as the **Velocity Specification Method** in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the **Velocity Magnitude**) and the direction of the vector:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the **Cartesian** coordinate system, you will define the **X**, **Y**, and (in 3D) **Z-Component of Flow Direction**.
- If your geometry is 2D axisymmetric, or you chose in step 3 to use a **Cylindrical** coordinate system, enter the appropriate values of **Radial**, **Axial**, and (if you are modeling axisymmetric swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.
- If you chose in step 3 to use a **Local Cylindrical** coordinate system, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

Figure 7.13 (p. 261) shows the vector components for these different coordinate systems.

7.3.4.1.4. Setting the Velocity Magnitude Normal to the Boundary

If you selected **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the **Velocity Magnitude**).

7.3.4.1.5. Setting the Velocity Components

If you selected **Components** as the **Velocity Specification Method** in step 1 above, you will enter the components of the velocity vector at the inflow boundary as follows:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the Cartesian coordinate system, you will define the **X**, **Y**, and (in 3D) **Z-Velocity**.
- If your geometry is 2D axisymmetric without swirl, you will set the **Radial** and **Axial-Velocity**.
- If your model is 2D axisymmetric with swirl, you will set the **Axial**, **Radial**, and **Swirl-Velocity**, and (optionally) the **Angular Velocity**, as described below.
- If you chose in step 3 to use a **Cylindrical** coordinate system, you will set the **Radial**, **Tangential**, and **Axial-Velocity**, and (optionally) the **Angular Velocity**, as described below.
- If you chose in step 3 to use a **Local Cylindrical** coordinate system, you will set the **Radial**, **Tangential**, and **Axial-Velocity**, and (optionally) the **Angular Velocity**, as described below, and then specify the **X**, **Y**, and **Z** component of the **Axis Origin** and the **Axis Direction**.

Important

Remember that positive values for x, y, and z velocities indicate flow in the positive x, y, and z directions. If flow enters the domain in the negative x direction, for example, you will need to specify a negative value for the x velocity. The same holds true for the radial, tangential, and axial velocities. Positive radial velocities point radially out from the axis, positive axial velocities are in the direction of the axis vector, and positive tangential velocities are based on the right-hand rule using the positive axis.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.4.1.6. Setting the Angular Velocity

If you chose **Components** as the **Velocity Specification Method** in step 1 above, and you are modeling axisymmetric swirl, you can specify the inlet **Angular Velocity** Ω in addition to the **Swirl-Velocity**. Similarly, if you chose **Components** as the **Velocity Specification Method** and you chose in step 3 to use a **Cylindrical** or **Local Cylindrical** coordinate system, you can specify the inlet **Angular Velocity** Ω in addition to the **Tangential-Velocity**.

If you specify Ω, v_θ is computed for each face as Ωr, where r is the radial coordinate in the coordinate system defined by the rotation axis and origin. If you specify both the **Swirl-Velocity** and the **Angular Velocity**, or the **Tangential-Velocity** and the **Angular Velocity**, ANSYS FLUENT will add v_θ and Ωr to get the swirl or tangential velocity at each face.

7.3.4.1.7. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the velocity inlet boundary conditions. Solution initialization is discussed in *Initializing the Solution* (p. 1239).

The **Supersonic/Initial Gauge Pressure** is ignored by ANSYS FLUENT whenever the flow is subsonic. If you choose to initialize the flow based on the velocity inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Remember that the static pressure value you enter is relative to the operating pressure set in the *Operating Conditions Dialog Box* (p. 1806). Note the comments in *Pressure Inputs and Hydrostatic Head* (p. 257)

7.3.4.1.8. Defining the Temperature

For calculations in which the energy equation is being solved, you will set the static temperature of the flow at the velocity inlet boundary in the **Thermal** tab in the **Temperature** field.

7.3.4.1.9. Defining Outflow Gauge Pressure

If you are using the density-based solver, you can specify an **Outflow Gauge Pressure** for a velocity inlet boundary. If the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the **Outflow Gauge Pressure** field.

7.3.4.1.10. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in *Determining Turbulence Parameters* (p. 250). Turbulence modeling in general is described in *Modeling Turbulence* (p. 643).

7.3.4.1.11. Defining Radiation Parameters

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See *Defining Boundary Conditions for Radiation* (p. 724) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.4.1.12. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see *Defining Cell Zone and Boundary Conditions for Species* (p. 822).

7.3.4.1.13. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in *Defining Non-Premixed Boundary Conditions* (p. 883).

7.3.4.1.14. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in *Setting Boundary Conditions for the Progress Variable* (p. 899).

7.3.4.1.15. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the velocity inlet. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.4.1.16. Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

7.3.4.2. Default Settings at Velocity Inlet Boundaries

Default settings (in SI) for velocity inlet boundary conditions are as follows:

Temperature	300
Velocity Magnitude	0
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
X-Velocity	0
Y-Velocity	0
Z-Velocity	0
Turbulent Kinetic Energy	1
Turbulent Dissipation Rate	1
Outflow Gauge Pressure	0

7.3.4.3. Calculation Procedure at Velocity Inlet Boundaries

ANSYS FLUENT uses your boundary condition inputs at velocity inlets to compute the mass flow into the domain through the inlet and to compute the fluxes of momentum, energy, and species through the inlet.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section describes these calculations for the case of flow entering the domain through the velocity inlet boundary and for the less common case of flow exiting the domain through the velocity inlet boundary.

7.3.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets

When your velocity inlet boundary condition defines flow entering the physical domain of the model, ANSYS FLUENT uses both the velocity components and the scalar quantities that you defined as boundary conditions to compute the inlet mass flow rate, momentum fluxes, and fluxes of energy and chemical species.

The mass flow rate entering a fluid cell adjacent to a velocity inlet boundary is computed as

$$\dot{m} = \int \rho \vec{v} \cdot d\vec{A} \qquad \textbf{(7–66)}$$

Note that only the velocity component normal to the control volume face contributes to the inlet mass flow rate.

7.3.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits

Sometimes a velocity inlet boundary is used where flow exits the physical domain. This approach might be used, for example, when the flow rate through one exit of the domain is known or is to be imposed on the model.

Important

In such cases you must ensure that overall continuity is maintained in the domain.

In the pressure-based solver, when flow exits the domain through a velocity inlet boundary ANSYS FLUENT uses the boundary condition value for the velocity component normal to the exit flow area. It does not use any other boundary conditions that you have input. Instead, all flow conditions except the normal velocity component are assumed to be those of the upstream cell.

In the density-based solver, if the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the **Outflow Gauge Pressure** field.

7.3.4.3.3. Density Calculation

Density at the inlet plane is either constant or calculated as a function of temperature, pressure, and/or species mass/mole fractions, where the mass or mole fractions are the values you entered as an inlet condition.

7.3.5. Mass Flow Inlet Boundary Conditions

Mass flow boundary conditions can be used in ANSYS FLUENT to provide a prescribed mass flow rate or mass flux distribution at an inlet. As with a velocity inlet, specifying the mass flux permits the total pressure to vary in response to the interior solution. This is in contrast to the pressure inlet boundary condition (see *Pressure Inlet Boundary Conditions* (p. 255)), where the total pressure is fixed while the mass flux varies. However, unlike a velocity inlet, the mass flow inlet is equally applicable to incompressible and compressible flows.

A mass flow inlet is often used when it is more important to match a prescribed mass flow rate than to match the total pressure of the inflow stream. An example is the case of a small cooling jet that is bled into the main flow at a fixed mass flow rate, while the velocity of the main flow is governed primarily by a (different) pressure inlet/outlet boundary condition pair. A mass flow inlet boundary condition can also be used as an outflow by specifying the flow direction away from the solution domain.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.5.1. Limitations and Special Considerations

- The adjustment of inlet total pressure might result in a slower convergence, so if both the pressure inlet boundary condition and the mass flow inlet boundary condition are acceptable choices, you should choose the former.
- It is not necessary to use mass flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow.
- A mass flow boundary operating as an outflow has the following limitations:
  - It is available for single-phase flow only.
  - It is not available with the VOF, mixture, and Eulerian multiphase models in the pressure-based solver.
  - It is not available with the Wet Steam model in the density-based solver.

For an overview of flow boundaries, see *Flow Inlet and Exit Boundary Conditions* (p. 249).

7.3.5.2. Inputs at Mass Flow Inlet Boundaries

7.3.5.2.1. Summary

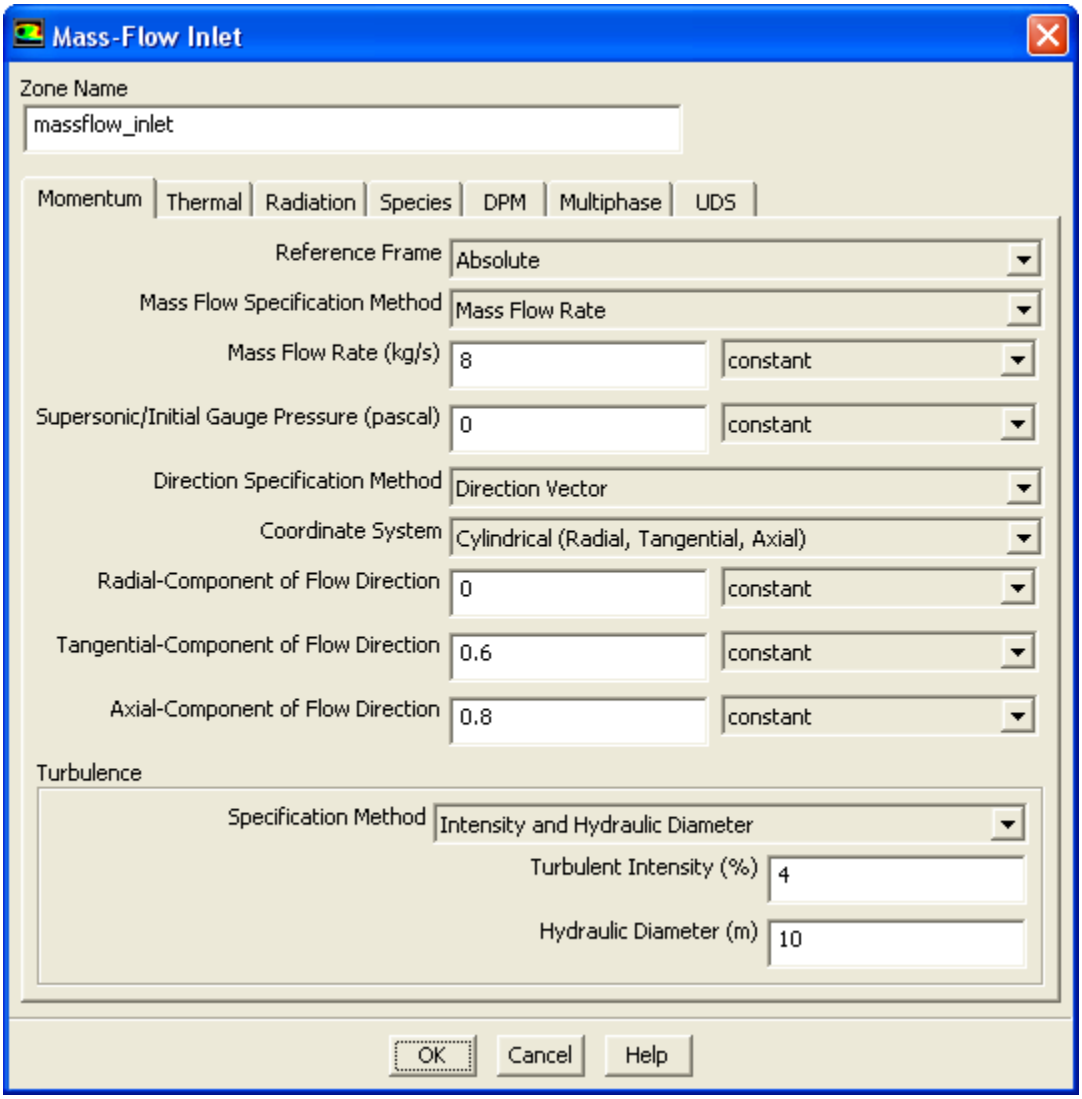
You will enter the following information for a mass flow inlet boundary:

- type of reference frame
- mass flow rate, mass flux, or (primarily for the mixing plane model) mass flux with average mass flux
- total (stagnation) temperature
- static pressure
- flow direction
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the *Mass-Flow Inlet Dialog Box* (p. 1832) (*Figure 7.15* (p. 272)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)). Note that open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.15 The Mass-Flow Inlet Dialog Box



7.3.5.2.2. Selecting the Reference Frame

You will have the option to specify the mass flow boundary conditions either in the absolute or relative reference frame, when the cell zone adjacent to the mass flow inlet is moving. For such a case, choose **Absolute** (the default) or **Relative to Adjacent Cell Zone** in the **Reference Frame** drop-down list. If the cell zone adjacent to the mass flow inlet is not moving, both formulations are equivalent.

7.3.5.2.3. Defining the Mass Flow Rate or Mass Flux

You can specify the mass flow rate through the inlet zone and have ANSYS FLUENT convert this value to mass flux, or specify the mass flux directly. For cases where the mass flux varies across the boundary, you can also specify an average mass flux; see below for more information about this specification method.

You can define the mass flux or mass flow rate using a profile or a user-defined function.

The inputs for mass flow rate or flux are as follows:

1. Specify the mass flow by selecting **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If you selected **Mass Flow Rate** (the default), set the prescribed mass flow rate in the **Mass Flow Rate** field when **constant** is selected from the drop-down list. Otherwise, select your hooked UDF or transient profile.

   Important

   The hooked UDF or transient profile can only be used to provide time-varying specification of mass flow rate. Therefore, the transient solver must be used to run the simulation. Note that the variation of profile with position in space is not applicable with this hookup.

   See DEFINE_PROFILE in the UDF Manual for an example of a mass flow inlet UDF.

   Important

   Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

3. If you selected **Mass Flux**, set the prescribed mass flux in the **Mass Flux** field, or select your hooked UDF or profile.
4. If you selected **Mass Flux with Average Mass Flux**, set the prescribed mass flux and average mass flux in the **Mass Flux** and **Average Mass Flux** fields.

7.3.5.2.4. More About Mass Flux and Average Mass Flux

As noted above, you can specify an average mass flux with the mass flux. If, for example, you specify a mass flux profile such that the average mass flux integrated over the zone area is 4.7, but you actually want to have a total mass flux of 5, you can keep the profile unchanged, and specify an average mass flux of 5. ANSYS FLUENT will maintain the profile shape but adjust the values so that the resulting mass flux across the boundary is 5.

The mass flux with average mass flux specification method is also used by the mixing plane model described in *The Mixing Plane Model* (p. 523). If the mass flow inlet boundary is going to represent one of the mixing planes, then you do *not* need to specify the mass flux or flow rate; you can keep the default **Mass Flow Rate** of 1. When you create the mixing plane later on in the problem setup, ANSYS FLUENT will automatically select the **Mass Flux with Average Mass Flux** method in the **Mass-Flow Inlet** dialog box and set the **Average Mass Flux** to the value obtained by integrating the mass flux profile for the upstream zone. This will ensure that mass is conserved between the upstream zone and the downstream (mass flow inlet) zone.

7.3.5.2.5. Defining the Total Temperature

Enter the value for the total (stagnation) temperature of the inflow stream in the **Total Temperature** field in the **Thermal** tab.

The total temperature is specified either in the absolute reference frame or relative to the adjacent cell zone, depending on your setting for the **Reference Frame**.

For the Eulerian multiphase model, the total temperature, and mass flux components need to be specified for the individual phases. The **Reference Frame** (**Relative to Adjacent Cell Zone** or **Absolute**) for each of the phases is the same as the reference frame selected for the mixture phase.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that you can only set the reference frame for the mixture, however, the total temperature can only be set for the individual phases.

Important

- If the flow is incompressible, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the static temperature.
- For the mixture multiphase model, if a boundary allows a combination of compressible and incompressible phases to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the static temperature at that boundary. If a boundary allows *only a compressible phase* to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is the total temperature (relative/absolute) at that boundary. The total temperature depends on the **Reference Frame** option selected in the **Mass-Flow Inlet** dialog box.
- For the VOF multiphase model, if a boundary allows a *compressible phase* to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the total temperature at that boundary. The total temperature (relative/absolute) depends on the **Reference Frame** option chosen in the dialog box. Otherwise, the temperature assigned to the boundary is considered to be the static temperature at the boundary.
- For the Eulerian multiphase model, if a boundary allows a mixture of compressible and incompressible phases in the domain, then the temperature of each of the phases is the total or static temperature, depending on whether the phase is compressible or incompressible. Total temperature (relative/absolute) depends on the **Reference Frame** option chosen in the **Mass-Flow Inlet** dialog box.

7.3.5.2.6. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in *Initializing the Solution* (p. 1239).

The **Supersonic/Initial Gauge Pressure** is ignored by ANSYS FLUENT whenever the flow is subsonic. If you choose to initialize the flow based on the mass flow inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Remember that the static pressure value you enter is relative to the operating pressure set in the *Operating Conditions Dialog Box* (p. 1806). Note the comments in *Pressure Inputs and Hydrostatic Head* (p. 257) regarding hydrostatic pressure.

7.3.5.2.7. Defining the Flow Direction

You can define the flow direction at a mass flow inlet explicitly, or you can define the flow to be normal to the boundary.

The procedure for defining the flow direction is as follows, referring to *Figure 7.15* (p. 272) :

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Specify the flow direction by selecting **Direction Vector**, **Normal to Boundary**, or **Outward Normals** in the **Direction Specification Method** drop-down list.
2. If you selected **Direction Vector** and your geometry is 2D, go to the next step. If your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** in the **Coordinate System** drop-down list. See *Defining the Flow Direction* (p. 259) for information about Cartesian, cylindrical, local cylindrical, and local cylindrical swirl coordinate systems.
3. If you selected **Direction Vector**, set the vector components as follows:
   - If your geometry is 2D non-axisymmetric, or you chose to use a 3D **Cartesian** coordinate system, enter appropriate values for the **X**, **Y**, and (in 3D) **Z-Component of Flow Direction**.
   - If your geometry is 2D axisymmetric, or you chose to use a 3D **Cylindrical** coordinate system, enter appropriate values for the **Axial**, **Radial**, and (if you are modeling swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.
   - If you chose to use a 3D **Local Cylindrical** coordinate system, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of **Axis Origin** and the **Axis Direction**.
   - If you chose to use a 3D **Local Cylindrical Swirl** coordinate system, enter appropriate values for the **Axial** and **Radial-Component of Flow Direction** in the axial and radial planes, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

   Important

   Local Cylindrical Swirl should not be used for open channel boundary conditions and on the mixing plane boundaries, while using the mixing plane model.

4. If you selected **Normal to Boundary**, there are no additional inputs for flow direction.

   Important

   Note that if you are modeling axisymmetric swirl, the flow direction will be normal to the boundary; i.e., there will be no swirl component at the boundary for axisymmetric swirl.

5. If **Outward Normals** is selected, then the mass flow boundary will operate as an outflow, pumping flow out of the domain with the rate specified in the **Mass Flow Specification Method**. If the mass flow rate is specified, then by default, the fluxes on the boundary will be allowed to vary to preserve the flow profile out of the domain. At convergence, the total mass flow rate should match the specified value. If constant mass flux is needed rather than the default variable fluxes to preserve the profiles, then you can do so via the text command `define /boundary-conditions/bc-settings/mass-flow`. Answer `no` when asked to `preserve profile while flow leaves`.

   Important

   The mass flow boundary can also operate as an outflow using the **Direction Vector** flow specification method if the flow components are pointing away from the boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.5.2.8. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in *Determining Turbulence Parameters* (p. 250). Turbulence modeling is described in *Modeling Turbulence* (p. 643).

7.3.5.2.9. Defining Radiation Parameters

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See *Defining Boundary Conditions for Radiation* (p. 724) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

7.3.5.2.10. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see *Defining Cell Zone and Boundary Conditions for Species* (p. 822).

7.3.5.2.11. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in *Defining Non-Premixed Boundary Conditions* (p. 883).

7.3.5.2.12. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in *Setting Boundary Conditions for the Progress Variable* (p. 899).

7.3.5.2.13. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the mass flow inlet. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.5.2.14. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See *Modeling Open Channel Flows* (p. 1121) for details.

7.3.5.3. Default Settings at Mass Flow Inlet Boundaries

Default settings (in SI) for mass flow inlet boundary conditions are as follows:

Mass Flow-Rate	1
Total Temperature	300
Supersonic/Initial Gauge Pressure	0
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Kinetic Energy	1
Turbulent Dissipation Rate	1

7.3.5.4. Calculation Procedure at Mass Flow Inlet Boundaries

When mass flow boundary conditions are used for an inlet zone, a velocity is computed for each face in that zone, and this velocity is used to compute the fluxes of all relevant solution variables into the domain. With each iteration, the computed velocity is adjusted so that the correct mass flow value is maintained.

To compute this velocity, your inputs for mass flow rate, flow direction, static pressure, and total temperature are used.

There are two ways to specify the mass flow rate. The first is to specify the total mass flow rate, $\dot{m}$, for the inlet. The second is to specify the mass flux, ρv_n (mass flow rate per unit area). If a total mass flow rate is specified, ANSYS FLUENT converts it internally to a uniform mass flux by dividing the mass flow rate by the total inlet area:

$$\rho v_n = \frac{\dot{m}}{A} \tag{7–67}$$

If the direct mass flux specification option is used, the mass flux can be varied over the boundary by using profile files or user-defined functions. If the average mass flux is also specified (either explicitly by you or automatically by ANSYS FLUENT), it is used to correct the specified mass flux profile, as described earlier in this section.

Once the value of ρv_n at a given face has been determined, the density, ρ, at the face must be determined in order to find the normal velocity, v_n. The manner in which the density is obtained depends upon whether the fluid is modeled as an ideal gas or not. Each of these cases is examined below.

7.3.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases

If the fluid is an ideal gas, the static temperature and static pressure are required to compute the density:

$$p = \rho R T \tag{7–68}$$

If the inlet is supersonic, the static pressure used is the value that has been set as a boundary condition. If the inlet is subsonic, the static pressure is extrapolated from the cells inside the inlet face.

The static temperature at the inlet is computed from the total enthalpy, which is determined from the total temperature that has been set as a boundary condition. The total enthalpy is given by

$$h_0(T_0) = h(T) + \frac{1}{2}v^2 \tag{7–69}$$

where the velocity magnitude is related to the mass flow rate given by *Equation 7–67* (p. 277) *Equation 7–68* (p. 277) to relate density to the (known) static pressure and (unknown) temperature, *Equation 7–69* (p. 277) can be solved to obtain the static temperature.

When the mass flow is used as an outflow with the profile preserving feature, a scaling factor of the specified mass flow rate over the computed mass flow rate at the boundary is used to scale the normal face velocities at the boundary. The other velocity components will be extrapolated from the interior. Flow variables such

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as pressure, temperature, species, or other scalar quantities will be also extrapolated from adjacent cell centers.

7.3.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows

When you are modeling incompressible flows, the static temperature is equal to the total temperature. The density at the inlet is either constant or readily computed as a function of the temperature and (optionally) the species mass or mole fractions. The velocity is then computed using *Equation 7–67* (p. 277).

7.3.5.4.3. Flux Calculations at Mass Flow Boundaries

To compute the fluxes of all variables at the inlet, the flux velocity, v_n, is used along with the inlet value of the variable in question. For example, the flux of mass is ρv_n, and the flux of turbulence kinetic energy is $\rho k v_n$. These fluxes are used as boundary conditions for the corresponding conservation equations during the course of the solution.

7.3.6. Inlet Vent Boundary Conditions

Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) pressure and temperature.

7.3.6.1. Inputs at Inlet Vent Boundaries

You will enter the following information for an inlet vent boundary:

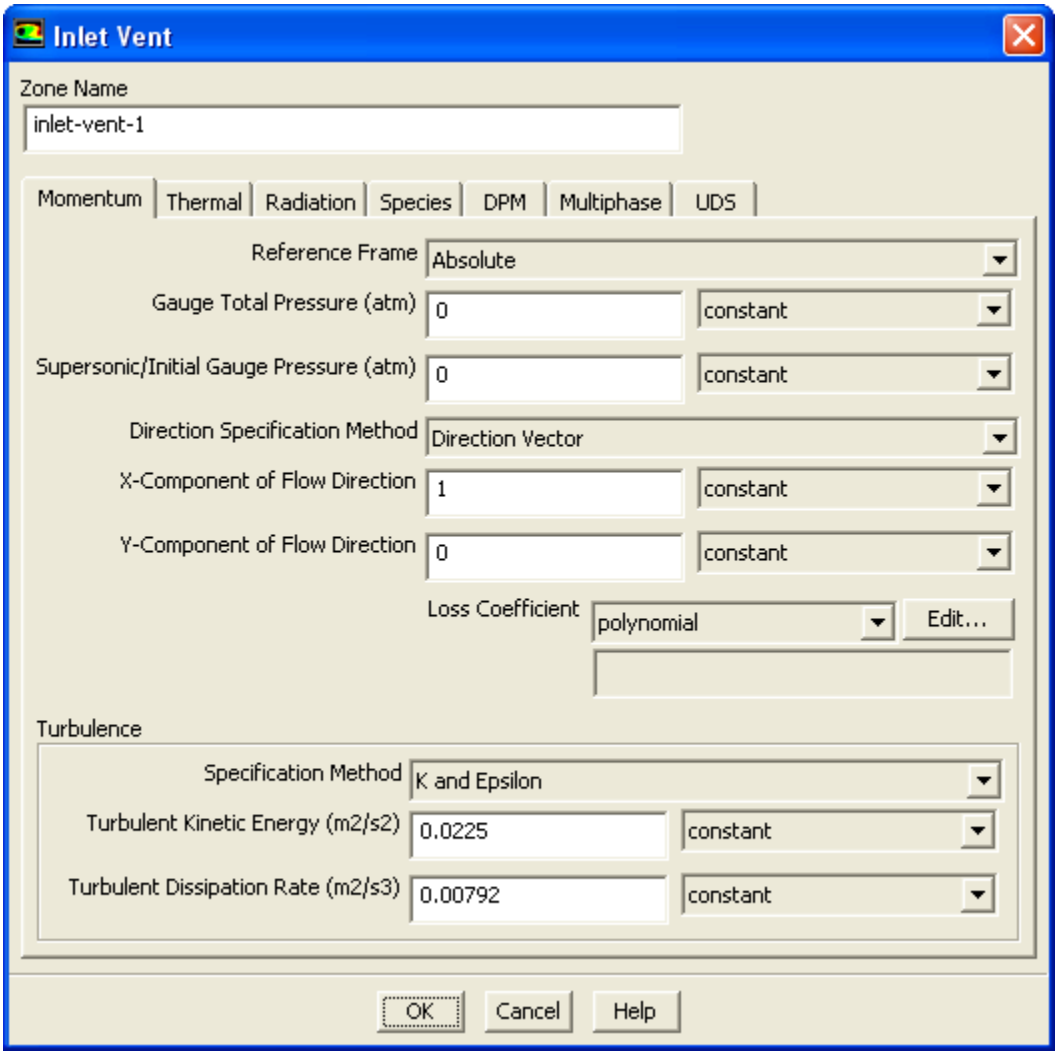
- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- loss coefficient
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the *Inlet Vent Dialog Box* (p. 1821) (*Figure 7.16* (p. 279)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

The first 12 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See *Inputs at Pressure Inlet Boundaries* (p. 255) for details. Specification of the loss coefficient is described here. Open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.16 The Inlet Vent Dialog Box



7.3.6.1.1. Specifying the Loss Coefficient

An inlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the vent, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{7–70}$$

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient.

Important

Δp is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can define the **Loss-Coefficient** across the vent as a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of the normal velocity. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details.

7.3.7. Intake Fan Boundary Conditions

Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) pressure and temperature.

7.3.7.1. Inputs at Intake Fan Boundaries

You will enter the following information for an intake fan boundary:

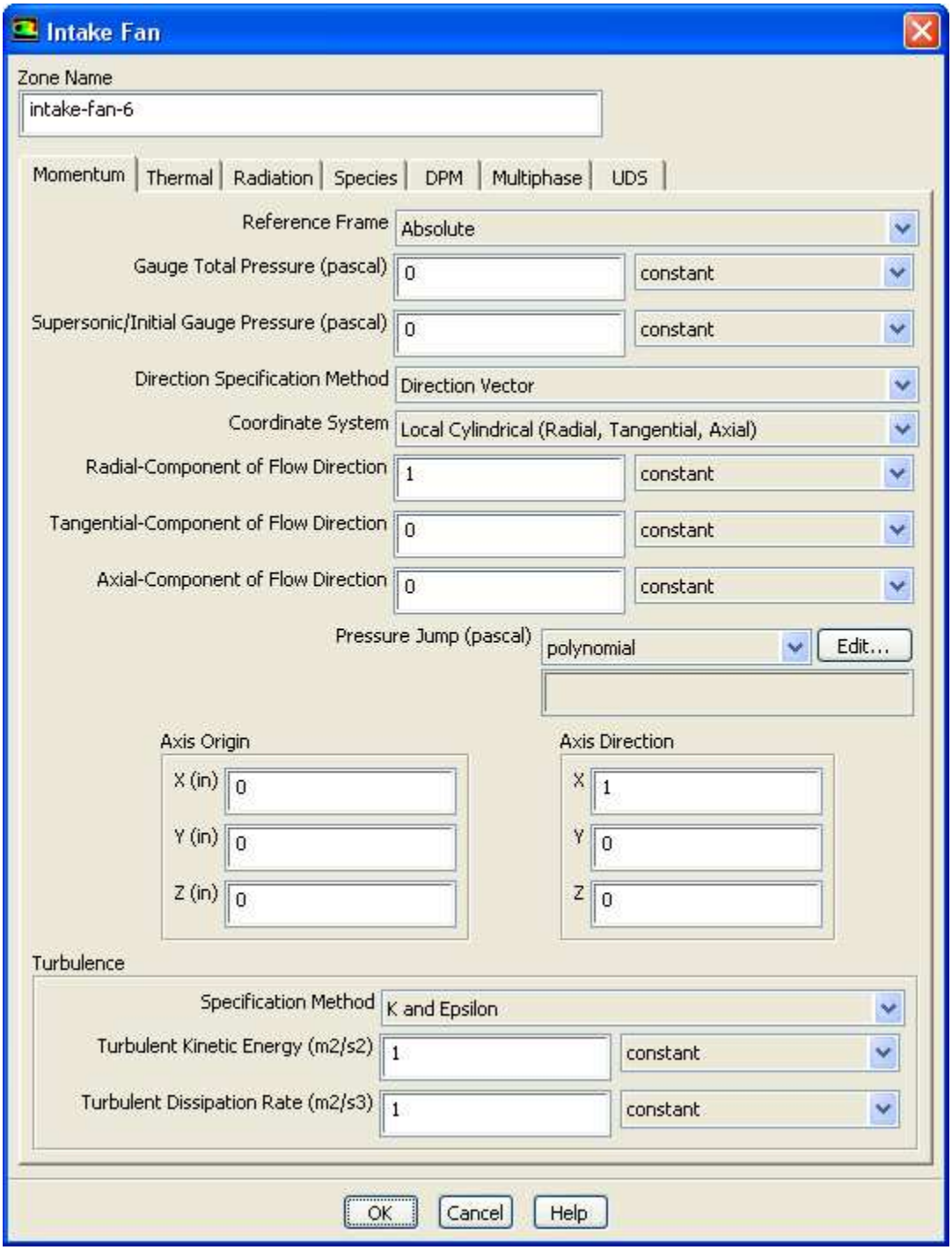
- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- pressure jump
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the *Intake Fan Dialog Box* (p. 1826) (shown in *Figure 7.17* (p. 281)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

The first 12 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See *Inputs at Pressure Inlet Boundaries* (p. 255) for details. Specification of the pressure jump is described here. Open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.17 The Intake Fan Dialog Box



7.3.7.1.1. Specifying the Pressure Jump

An intake fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. In the case of reversed flow, the fan is treated like an outlet vent with a loss coefficient of unity.

You can define the **Pressure-Jump** across the fan as a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of the normal velocity. The dialog boxes for defining these functions are the same as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

those used for defining temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details.

7.3.8. Pressure Outlet Boundary Conditions

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of the specified static pressure is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure will no longer be used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior.

A set of "backflow" conditions is also specified should the flow reverse direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities.

Several options in ANSYS FLUENT exist, where a radial equilibrium outlet boundary condition can be used (see *Defining Static Pressure* (p. 283) and a target mass flow rate for pressure outlets (see *Target Mass Flow Rate Option* (p. 288) for details) can be specified.

For an overview of flow boundaries, see *Flow Inlet and Exit Boundary Conditions* (p. 249).

7.3.8.1. Inputs at Pressure Outlet Boundaries

7.3.8.1.1. Summary

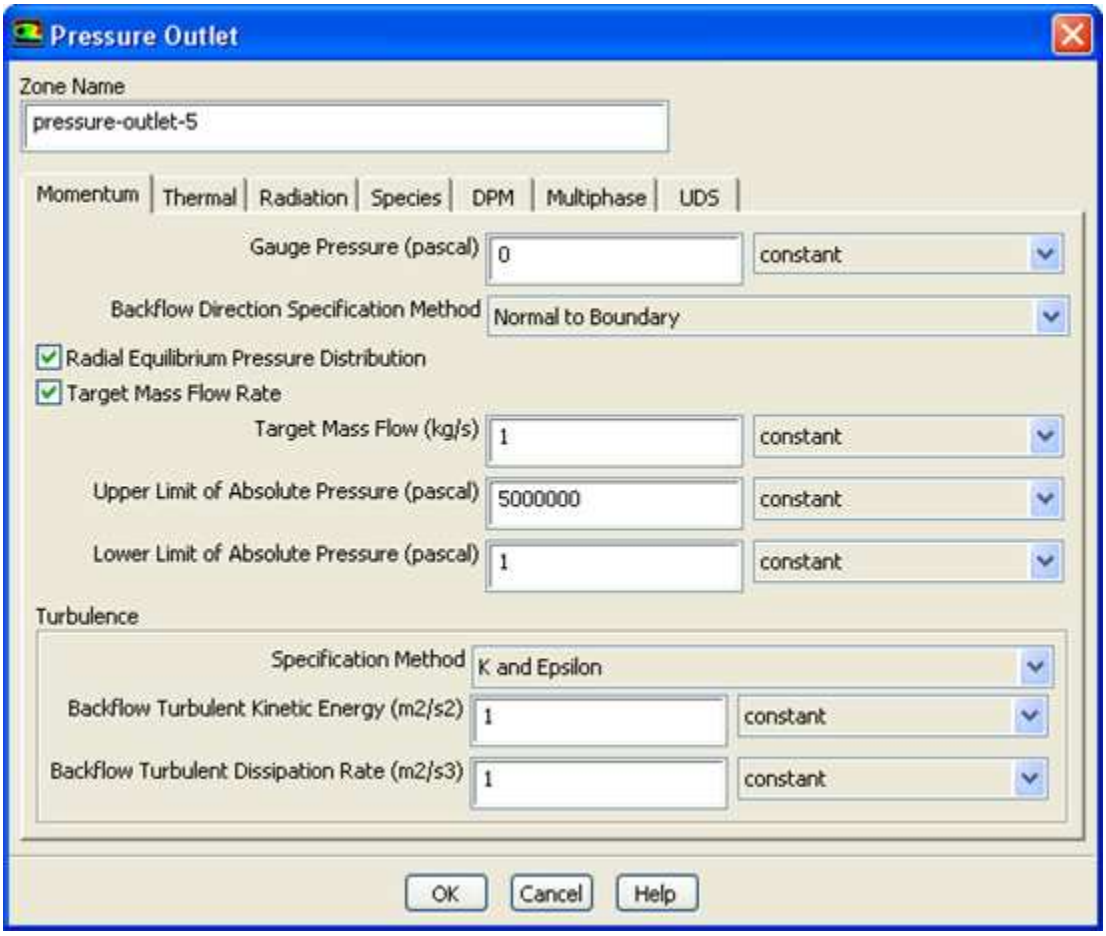
You will enter the following information for a pressure outlet boundary:

- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- discrete phase boundary conditions (for discrete phase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- non-reflecting boundary (for compressible density-based solver, see *General Non-Reflecting Boundary Conditions* (p. 351) for details)
- target mass flow rate (not available for multiphase flows)

All values are entered in the *Pressure Outlet Dialog Box* (p. 1853) (*Figure 7.18* (p. 283)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)). Note that open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.18 The Pressure Outlet Dialog Box



7.3.8.1.2. Defining Static Pressure

To set the static pressure at the pressure outlet boundary, enter the appropriate value for **Gauge Pressure** in the **Pressure Outlet** dialog box. This value will be used for subsonic flow only. Should the flow become locally supersonic, the pressure will be extrapolated from the upstream conditions.

Remember that the static pressure value you enter is relative to the operating pressure set in the *Operating Conditions Dialog Box* (p. 1806). Refer to *Pressure Inputs and Hydrostatic Head* (p. 257) regarding hydrostatic pressure.

ANSYS FLUENT also provides an option to use a radial equilibrium outlet boundary condition. To enable this option, turn on **Radial Equilibrium Pressure Distribution**. When this feature is active, the specified gauge pressure applies only to the position of minimum radius (relative to the axis of rotation) at the boundary. The static pressure on the rest of the zone is calculated from the assumption that radial velocity is negligible, so that the pressure gradient is given by

$$\frac{\partial p}{\partial r} = \frac{\rho v_\theta^2}{r} \tag{7–71}$$

where r is the distance from the axis of rotation and v_θ is the tangential velocity. Note that this boundary condition can be used even if the rotational velocity is zero. For example, it could be applied to the calculation of the flow through an annulus containing guide vanes.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the radial equilibrium outlet condition is available only for 3D and axisymmetric swirl calculations.

ANSYS FLUENT also provides an option to use an **Average Pressure Specification** method at the pressure outlet when using the pressure-based solver. This option allows the pressure along the outlet boundary to vary, but maintain an average equivalent to the specified value in the **Gauge Pressure** input field. In this boundary implementation, the pressure variation provides a low level of non-reflectivity. For more details, see *Calculation Procedure at Pressure Outlet Boundaries* (p. 285).

Note

The **Average Pressure Specification** option is not available if the **Radial Equilibrium Pressure Distribution** option is enabled.

7.3.8.1.3. Defining Backflow Conditions

Backflow properties consistent with the models you are using will appear in the **Pressure Outlet** dialog box. The specified values will be used only if flow is pulled in through the outlet.

- The **Backflow Total Temperature** (in the **Thermal** tab) should be set for problems involving energy calculation.
- When the direction of the backflow re-entering the computational domain is known, and deemed to be relevant to the flow field solution, you can specify it choosing one of the options available in the **Backflow Direction Specification Method** drop-down list. The default value for this field is **Normal to Boundary**, and requires no further input. If you choose **Direction Vector**, the dialog box will expand to show the inputs for the components of the direction vector for the backflow, and if you are running the 3D version of ANSYS FLUENT, the dialog box will display a **Coordinate System** drop-down list. If you choose **From Neighboring Cell**,ANSYS FLUENT will determine the direction of the backflow using the direction of the flow in the cell layer adjacent to the pressure outlet.
- For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use in determining appropriate values for these inputs are provided in *Determining Turbulence Parameters* (p. 250). Turbulence modeling in general is described in *Modeling Turbulence* (p. 643).
- If you are modeling species transport, you will set the backflow species mass or mole fractions under **Species Mass Fractions** or **Species Mole Fractions**. For details, see *Defining Cell Zone and Boundary Conditions for Species* (p. 822).
- If you are modeling combustion using the non-premixed or partially premixed combustion model, you will set the backflow mixture fraction and variance values. See *Defining Non-Premixed Boundary Conditions* (p. 883) for details.
- If you are modeling combustion using the premixed or partially premixed combustion model, you will set the backflow **Progress Variable** value. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.
- If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If backflow occurs, the pressure you specified as the **Gauge Pressure** will be used as total pressure, so you need not specify a backflow pressure value explicitly. The flow direction will be based on your specification of the direction vector.

  If the cell zone adjacent to the pressure outlet is moving (i.e., if you are using a moving reference frame, multiple reference frames, mixing planes, or sliding meshes) and you are using the pressure-based solver, the velocity in the dynamic contribution to total pressure (see *Equation 7–57* (p. 258)) will be absolute or relative to the motion of the cell zone, depending on whether or not the **Absolute** velocity formulation is enabled in the **General** task page. For the density-based solver, the velocity in *Equation 7–57* (p. 258) (or the Mach number in *Equation 7–58* (p. 258)) is always in the absolute frame.

Important

Even if no backflow is expected in the converged solution, you should always set realistic values to minimize convergence difficulties in the event that backflow does occur during the calculation.

7.3.8.1.4. Defining Radiation Parameters

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the **Internal Emissivity** and (optional) **External Black Body Temperature Method**. See *Defining Boundary Conditions for Radiation* (p. 724) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

7.3.8.1.5. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure outlet. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.8.1.6. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See *Modeling Open Channel Flows* (p. 1121) for details.

7.3.8.2. Default Settings at Pressure Outlet Boundaries

Default settings (in SI) for pressure outlet boundary conditions are as follows:

Gauge Pressure	0
Backflow Total Temperature	300
Backflow Turbulent Kinetic Energy	1
Backflow Turbulent Dissipation Rate	1

7.3.8.3. Calculation Procedure at Pressure Outlet Boundaries

At pressure outlets, ANSYS FLUENT uses the boundary condition pressure you input as the static pressure of the fluid at the outlet plane, P_s, and extrapolates all other conditions from the interior of the domain.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.8.3.1. Pressure-Based Solver Implementation

In the pressure-based solver, by default, the face pressure at the boundary is the same as the value specified in the **Pressure Outlet** dialog box. When the **Average Pressure Specification** option is enabled, the face pressure value P_f for subsonic flows at the outlet boundary is computed using the following expressions:

$$P_f = 0.5\,(P_c + P_e) + dp \tag{7–72}$$

where

P_c = interior cell pressure at neighboring exit face, f

P_e = specified exit pressure

dp = difference in the pressure value between the specified pressure P_e and the latest average pressure for the boundary defined by *Equation 7–73* (p. 286)

$$dp = \left(P_e - \frac{\sum_{i=1}^{i=n\_face} 0.5\,(P_c + P_e)\,(Area)}{\sum_{i=1}^{i=n\_face} Area} \right) \tag{7–73}$$

Typically, this averaging option should be used when the flow at the boundary exit is subsonic. If the flow is fully supersonic, then the **Average Pressure Specification** has no effect. If the flow at the exit is partially supersonic, then the face pressure value at the supersonic flow location will be extrapolated from the interior cells and will be included in the averaging procedure.

Note

If a geometric opening was modeled as a multiple pressure-outlet boundary condition with the average pressure specification, then the averaging is applied on each individual boundary condition and not over the entire geometric opening.

The limitations that exist when using the **Average Pressure Specification** option are listed below:

- The **Average Pressure Specification** option is not available when **Radial Equilibrium Pressure Distribution** is enabled.
- If the pressure outlet boundary is part of the mixing plane model pair, then the **Average Pressure Specification** option will not be available for that particular boundary zone.
- The **Average Pressure Specification** option is not available with multiphase flows.
- If a profile is specified for the **Gauge Pressure** field instead of a constant value, then the **Average Pressure Specification** will not be applied.

7.3.8.3.2. Density-Based Solver Implementation

In the density-based solver, there are three pressure specification methods available:

1. Weak enforcement of average pressure (default)
2. Strong enforcement of average pressure
3. Direct pressure specification

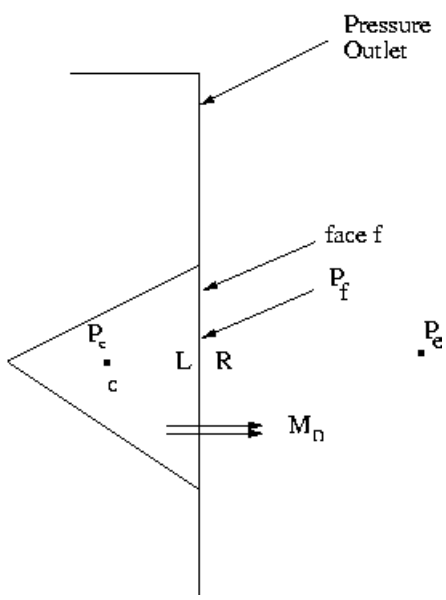
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The specification methods can be changed from the text user interface:

```
define/boundary-conditions/bc-settings/pressure-outlet
```

In the direct pressure specifications, the face pressure at the boundary is same as the value specified in the **Pressure Outlet** dialog box. The implementation is similar to that in the pressure-based solver. However, the default specification method in the density-based solver is the weak enforcement of average pressure. In this implementation for subsonic flow, the pressure at the faces of the outlet boundary is computed using a weighted average of the left and right state of the face boundary. This weighting is a blend of fifth-order polynomials based on the exit face normal Mach number [*46*] (p. 2211). Therefore, the face pressure P_f is a function of (P_c, P_e, M_n), where P_c is the interior cell pressure neighboring the exit face f, P_e is the specified exit pressure, and M_n is the face normal Mach number.

Figure 7.19 Pressures at the Face of a Pressure Outlet Boundary



For incompressible flows, the face pressure is computed as an average between the specified pressure and the interior pressure.

$$P_f = 0.5\,(P_c + P_e) \tag{7–74}$$

In this boundary implementation, the exit pressure is not constant along the pressure outlet boundary. However, upon flow convergence, the average boundary pressure will be close to the specified static exit pressure.

In general the weak average pressure enforcement works well in most flow situations. However, for cases where the computed average pressure value does not match the specified pressure value at the boundary (typically this happen when we have a coarse mesh and stretched cells near the pressure-outlet boundary) then the strong average pressure enforcement can be used to guarantee the specified pressure equal to the boundary average pressure. The strong enforcement is achieved by adding locally the difference in pressure value between the latest average pressure for the boundary and the face pressure obtained from weak enforcement. The strong enforcement is applicable when the flow is fully subsonic throughout the boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For all of the three pressure specification methods, if the flow becomes locally supersonic, then the face pressure values P_f are extrapolated from the interior cell pressure.

Important

When one of the NRBC model, none of the above specification methods are relevant since face pressure will be obtained from special NRBC procedures.

Important

If you are specifying a profile rather than a constant value for exit pressure, then you should not use this weak or strong enforcement of average pressure boundary. Instead, you should use the direct pressure specification method.

7.3.8.4. Other Optional Inputs at Pressure Outlet Boundaries

7.3.8.4.1. Non-Reflecting Boundary Conditions Option

One of the options that may be used at pressure outlets is non-reflecting boundary conditions (NRBC option is used when waves are made to pass through the boundaries while avoiding false reflections. Details of non-reflecting boundary conditions can be found in *General Non-Reflecting Boundary Conditions* (p. 351) of this chapter.

7.3.8.4.2. Target Mass Flow Rate Option

The simple Bernoulli's equation is used to adjust the pressure at every iteration on a pressure outlet zone in order to meet the desired mass flow rate. The change in pressure, based on Bernoulli's equation is given by the following equation:

$$dP = 0.5\rho_{ave}\left(\dot{m}^2 - \dot{m}_{req}^2\right) / \left(\rho_{ave} A\right)^2 \qquad \textbf{(7–75)}$$

where dP is the change in pressure, $\dot{m}$ is the current computed mass flow rate at the pressure-outlet boundary, $\dot{m}_{req}$ is the required mass flow rate, ρ_{ave} is the computed average density at the pressure-outlet boundary, and A is the area of the pressure-outlet boundary.

7.3.8.4.3. Limitations

- The target mass flow rate option is not available with multiphase flows or when any of the non-reflecting boundary conditions models are used.
- If the pressure-outlet zone is used in the mixing-plane model, the target mass flow rate option will not be available for that particular zone.
- The pressure outlet will not achieve the target mass flow rate if the flow becomes choked (i.e., the Mach number of the fluid in the pressure-outlet zone becomes equal to 1).

7.3.8.4.4. Target Mass Flow Rate Settings

To use the target mass flow rate option

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable **Target Mass Flow Rate** in the **Pressure Outlet** dialog box.
2. Specify the **Target Mass Flow** as either a constant value or hook a UDF to set the target mass flow rate.

   The settings for the target mass flow rate option can be accessed from the `target-mass-flow-rate-settings` text command:

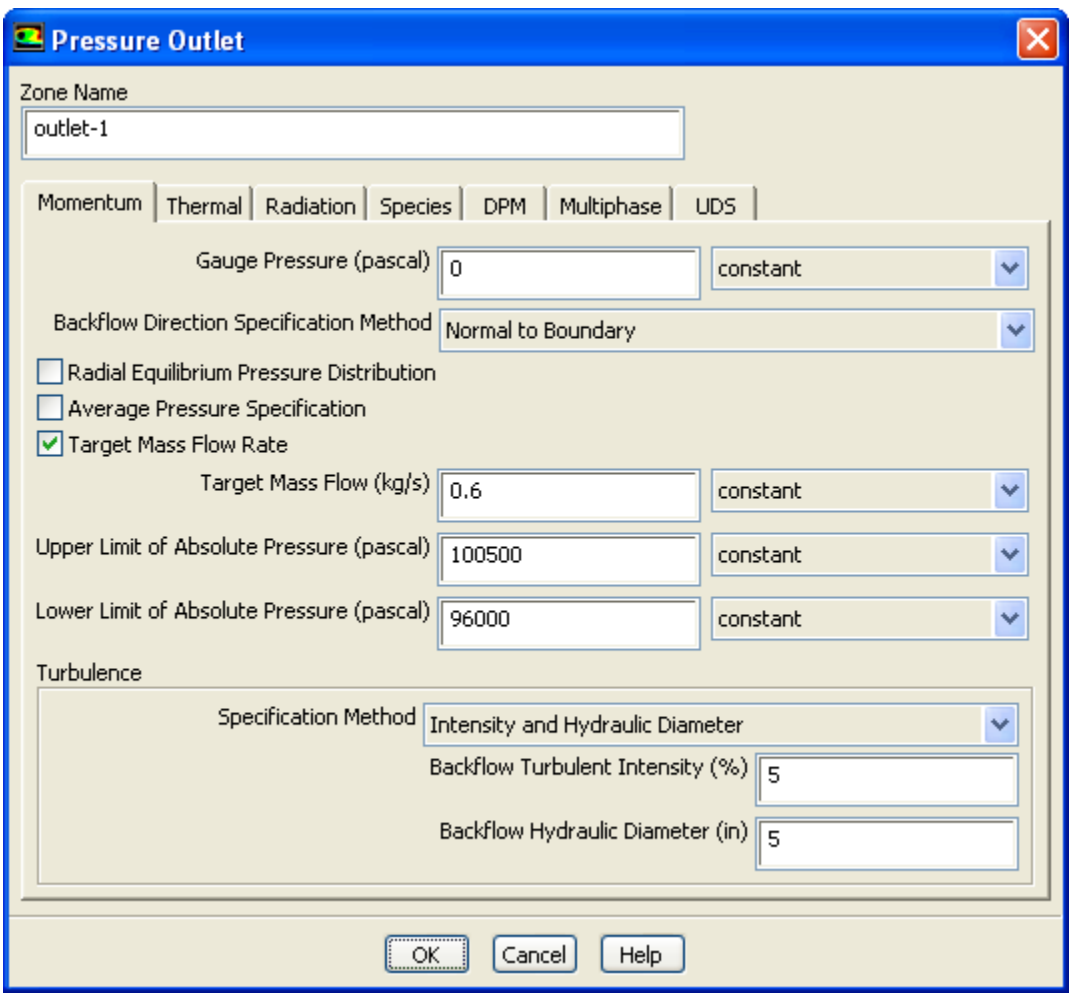
   `define` → `boundary-conditions` → `target-mass-flow-rate-settings`

   You will be prompted to

   a. Set the under-relaxation factor (the default setting is 0.05).

   b. Enable the targeted mass flow rate verbosity (the default is `no`). If enabled, it prints to the console window the required mass flow rate, computed mass flow rate, mean pressure, the new pressure imposed on the outlet, and the change in pressure in SI units.
3. In the **Pressure Outlet** dialog box, specify the **Upper Limit of Absolute Pressure** and **Lower Limit of Absolute Pressure**. Specifying the range of the pressure limits improves convergence in cases with a large number of outlet boundaries, which have different pressure variations on different boundaries. You can also use the `define/boundary-conditions/pressure-outlet` text command to specify these limits.

Figure 7.20 The Pressure Outlet Dialog Box with the Target Mass Flow Rate Option Enabled



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.8.4.5. Solution Strategies When Using the Target Mass Flow Rate Option

If convergence difficulties are encountered or if the solution is not converging at the desired mass flow rate, then try to lower the under-relaxation factor from the default value. Otherwise, you can use the alternate method to converge at the required mass flow rate.

In some cases, you may want to switch off the target mass flow rate option initially, then guess an exit pressure that will bring the solution closer to the target mass flow rate. After the solution stabilizes, you can turn on the target mass flow rate option and iterate to convergence. For many complex flow problems, this strategy is usually very successful.

The use of Full Multigrid Initialization is also very helpful in obtaining a good starting solution and in general will reduce the time required to get a converged solution on a target mass flow rate. For further information on Full Multigrid Initialization, see *Full Multigrid (FMG) Initialization* (p. 1244).

7.3.8.4.6. Setting Target Mass Flow Rates Using UDFs

For some unsteady problems it is desirable that the target mass flow rate be a function of the physical flow time. This enforcement of boundary condition can be done by attaching a UDF with `DEFINE_PROFILE` functions to the target mass flow rate field.

Important

Note that the mass flow rate profile is a function of time and only one constant value should be applied to all zone faces at a given time.

An example of a simple UDF using a `DEFINE_PROFILE` that will adjust the mass flow rate can be found in DEFINE_PROFILE in the UDF Manual.

7.3.9. Pressure Far-Field Boundary Conditions

Pressure far-field conditions are used in ANSYS FLUENT to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information (Riemann invariants) to determine the flow variables at the boundaries.

7.3.9.1. Limitations

Note the following limitations and restrictions when using pressure far-field boundary conditions:

- This boundary condition is applicable only when the density is calculated using the ideal-gas law (see *Density* (p. 403)). Using it for other flows is not permitted. To effectively approximate true infinite-extent conditions, you must place the far-field boundary far enough from the object of interest. For example, in lifting airfoil calculations, it is not uncommon for the far-field boundary to be a circle with a radius of 20 chord lengths.
- It is incompatible with the multiphase models (VOF, mixture, and Eulerian) that are available with the pressure-based solver.
- It cannot be applied to flows that employ constant density, the real gas model, and the wet steam model, which are available in the density-based solver.

For an overview of flow boundaries, see *Flow Inlet and Exit Boundary Conditions* (p. 249).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.9.2. Inputs at Pressure Far-Field Boundaries

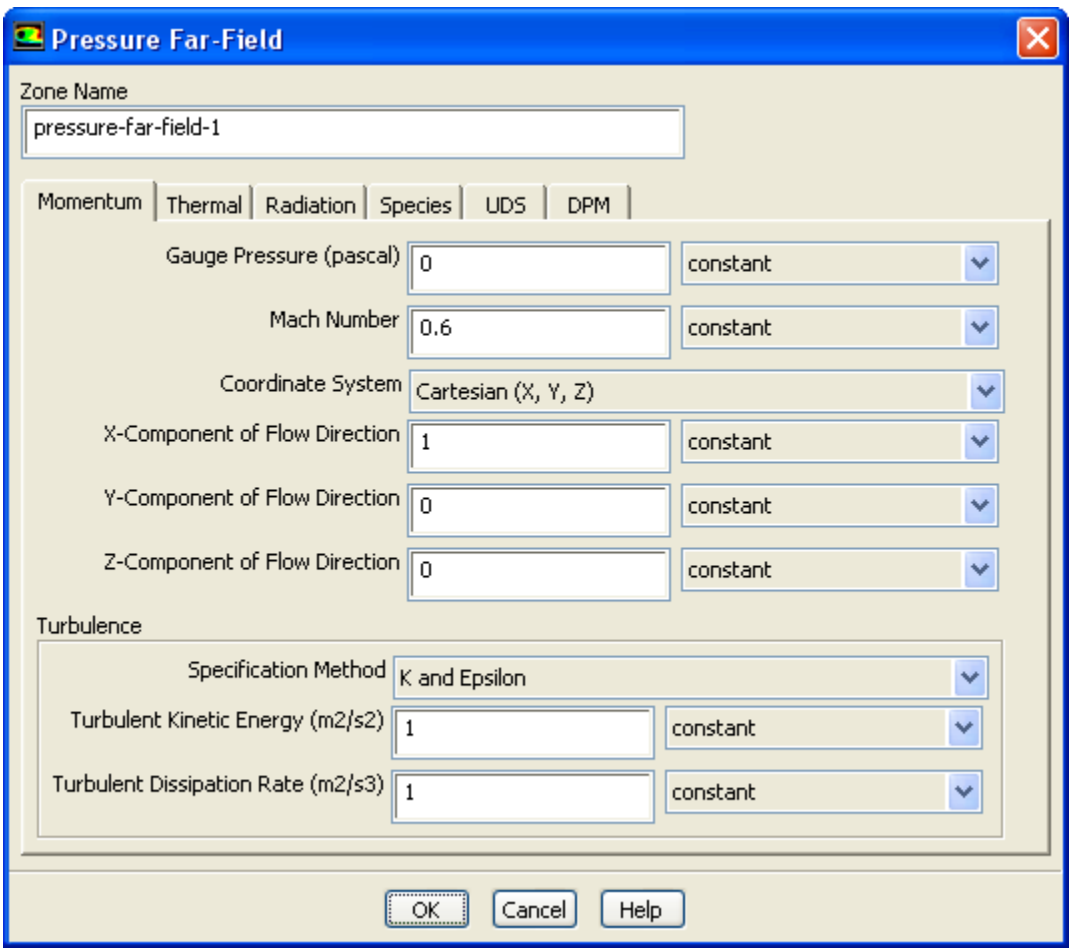
7.3.9.2.1. Summary

You will enter the following information for a pressure far-field boundary:

- static pressure
- Mach number
- temperature
- flow direction
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- chemical species mass or mole fractions (for species calculations)
- discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the *Pressure Far-Field Dialog Box* (p. 1844) (*Figure 7.21* (p. 291)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Figure 7.21 The Pressure Far-Field Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.9.2.2. Defining Static Pressure, Mach Number, and Static Temperature

To set the static pressure and temperature at the far-field boundary in the **Pressure Far-Field** dialog box, enter the appropriate values for **Gauge Pressure** and **Mach Number** in the **Momentum** tab. The Mach number can be subsonic, sonic, or supersonic. Set the **Temperature** in the **Thermal** tab.

7.3.9.2.3. Defining the Flow Direction

You can define the flow direction at a pressure far-field boundary by setting the components of the direction vector. If your geometry is 2D non-axisymmetric enter appropriate values for **X** and **Y** in the **Pressure Far-Field** dialog box (*Figure 7.21* (p. 291)). If your geometry is 2D axisymmetric, enter the appropriate values for **Axial**, **Radial**, and (if you are modeling axisymmetric swirl) **Tangential-Component of Flow Direction**.

If your geometry is 3D, you can choose a **Coordinate System** that is **Cartesian**, **Cylindrical**, or **Local Cylindrical**. In the Cartesian coordinate system, enter the appropriate values for **X**, **Y**, and **Z-Component of Flow Direction**. If the direction cosine data on the boundary is available, then use the cylindrical or local cylindrical coordinate system and specify the **Axial**, **Radial**, **Tangential-Component of Flow Direction**. For **Cylindrical**, axis parameters need to be specified on the adjacent cell zone of the boundary face. For **Local Cylindrical Swirl**, specify the **Axis Origin** and **Axis Direction**.

7.3.9.2.4. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in *Determining Turbulence Parameters* (p. 250). Turbulence modeling is described in *Modeling Turbulence* (p. 643).

7.3.9.2.5. Defining Radiation Parameters

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature Method**. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

7.3.9.2.6. Defining Species Transport Parameters

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mass Fractions** or **Species Mole Fractions**. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details.

7.3.9.3. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure far-field boundary. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.9.4. Default Settings at Pressure Far-Field Boundaries

Default settings (in SI) for pressure far-field boundary conditions are as follows:

Gauge Pressure	0
Mach Number	0.6
Temperature	300
X-Component of Flow Direction	1
Y-Component of Flow Direction	0

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z-Component of Flow Direction	0
Turbulent Kinetic Energy	1
Turbulent Dissipation Rate	1

7.3.9.5. Calculation Procedure at Pressure Far-Field Boundaries

The pressure far-field boundary condition is a non-reflecting boundary condition based on the introduction of Riemann invariants (i.e., characteristic variables) for a one-dimensional flow normal to the boundary. For flow that is subsonic there are two Riemann invariants, corresponding to incoming and outgoing waves:

$$R_\infty = v_{n_\infty} - \frac{2c_\infty}{\gamma - 1} \tag{7–76}$$

$$R_i = v_{n_i} + \frac{2c_i}{\gamma - 1} \tag{7–77}$$

where v_n is the velocity magnitude normal to the boundary, c is the local speed of sound and γ is the ratio of specific heats (ideal gas). The subscript ∞ refers to conditions being applied at infinity (the boundary conditions), and the subscript i refers to conditions in the interior of the domain (i.e., in the cell adjacent to the boundary face). These two invariants can be added and subtracted to give the following two equations:

$$v_n = \frac{1}{2}(R_i + R_\infty) \tag{7–78}$$

$$c = \frac{\gamma - 1}{4}(R_i - R_\infty) \tag{7–79}$$

where v_n and c become the values of normal velocity and sound speed applied on the boundary. At a face through which flow exits, the tangential velocity components and entropy are extrapolated from the interior; at an inflow face, these are specified as having free-stream values. Using the values for v_n, c, tangential velocity components, and entropy the values of density, velocity, temperature, and pressure at the boundary face can be calculated.

7.3.10. Outflow Boundary Conditions

Outflow boundary conditions in ANSYS FLUENT are used to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. You do not define any conditions at outflow boundaries (unless you are modeling radiative heat transfer, a discrete phase of particles, or split mass flow): ANSYS FLUENT extrapolates the required information from the interior. It is important, however, to understand the limitations of this boundary type.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that outflow boundaries cannot be used in the following cases:

- If a problem includes pressure inlet boundaries; use pressure outlet boundary conditions (see *Pressure Outlet Boundary Conditions* (p. 282)) instead.
- If you are modeling compressible flow.
- If you are modeling unsteady flows with varying density, even if the flow is incompressible.
- With the multiphase models (Eulerian, mixture, and VOF (except when modeling open channel flow, as described in Open Channel Flow n the Theory Guide).

For an overview of flow boundaries, see *Flow Inlet and Exit Boundary Conditions* (p. 249).

7.3.10.1. ANSYS FLUENT's Treatment at Outflow Boundaries

The boundary conditions used by ANSYS FLUENT at outflow boundaries are as follows:

- A zero diffusion flux for all flow variables.
- An overall mass balance correction.

The zero diffusion flux condition applied at outflow cells means that the conditions of the outflow plane are extrapolated from within the domain and have no impact on the upstream flow. The extrapolation procedure used by ANSYS FLUENT updates the outflow velocity and pressure in a manner that is consistent with a fully-developed flow assumption, as noted below, when there is no area change at the outflow boundary.

The zero diffusion flux condition applied by ANSYS FLUENT at outflow boundaries is approached physically in fully-developed flows. Fully-developed flows are flows in which the flow velocity profile (and/or profiles of other properties such as temperature) is unchanging in the flow direction.

It is important to note that gradients in the cross-stream direction may exist at an outflow boundary. Only the diffusion fluxes in the direction normal to the exit plane are assumed to be zero.

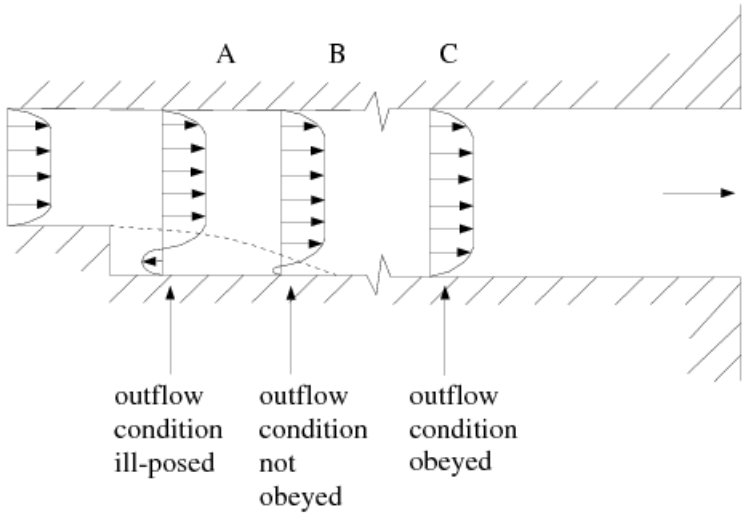
7.3.10.2. Using Outflow Boundaries

As noted in *ANSYS FLUENT's Treatment at Outflow Boundaries* (p. 294), the outflow boundary condition is obeyed in fully-developed flows where the diffusion flux for all flow variables in the exit direction are zero. However, you may also define outflow boundaries at physical boundaries where the flow is not fully developed—and you can do so with confidence if the assumption of a zero diffusion flux at the exit is expected to have a small impact on your flow solution. The appropriate placement of an outflow boundary is described by example below.

- Outflow boundaries where normal gradients are negligible: *Figure 7.22* (p. 295) shows a simple two-dimensional flow problem and several possible outflow boundary location choices. Location C shows the outflow boundary located upstream of the plenum exit but in a region of the duct where the flow is fully-developed. At this location, the outflow boundary condition is exactly obeyed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.22 Choice of the Outflow Boundary Condition Location



- Ill-posed outflow boundaries: Location B in *Figure 7.22* (p. 295) shows the outflow boundary near the reattachment point of the recirculation in the wake of the backward-facing step. This choice of outflow boundary condition is ill-posed as the gradients normal to the exit plane are quite large at this point and can be expected to have a significant impact on the flow field upstream. Because the outflow boundary condition ignores these axial gradients in the flow, location B is a poor choice for an outflow boundary. The exit location should be moved downstream from the reattachment point.

  Figure 7.22 (p. 295) shows a second ill-posed outflow boundary at location A. Here, the outflow is located where flow is pulled into the ANSYS FLUENT domain through the outflow boundary. In situations like this the ANSYS FLUENT calculation typically does not converge and the results of the calculation have no validity. This is because when flow is pulled into the domain through an outflow, the mass flow rate through the domain is "floating" or undefined. In addition, when flow enters the domain through an outflow boundary, the scalar properties of the flow are not defined. For example, the temperature of the flow pulled in through the outflow is not defined. (ANSYS FLUENT chooses the temperature using the temperature of the fluid adjacent to the outflow, inside the domain.) Thus you should view all calculations that involve flow entering the domain through an outflow boundary with skepticism. For such calculations, pressure outlet boundary conditions (see *Pressure Outlet Boundary Conditions* (p. 282)) are recommended.

Important

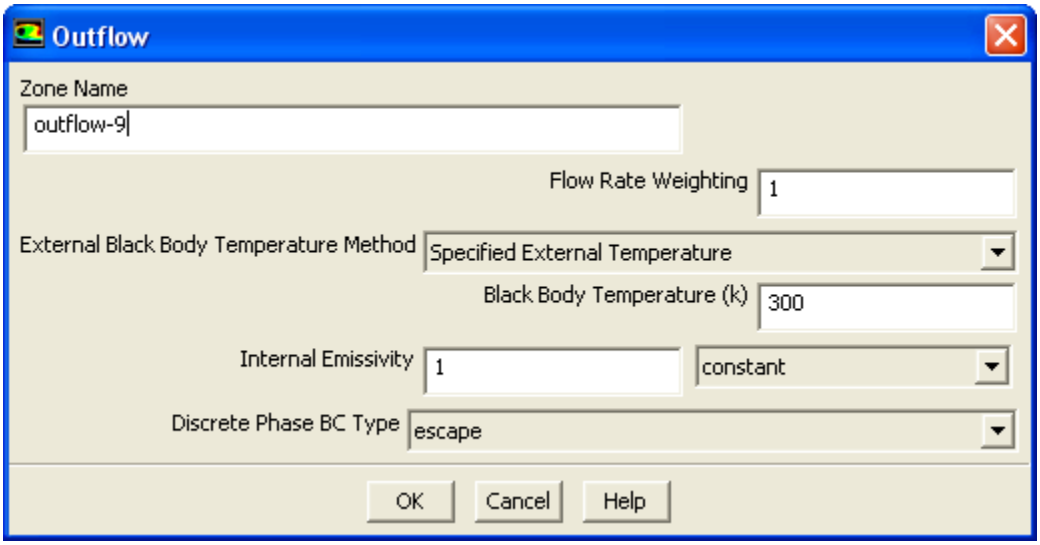
Note that convergence may be affected if there is recirculation through the outflow boundary at any point during the calculation, even if the final solution is not expected to have any flow reentering the domain. This is particularly true of turbulent flow simulations.

7.3.10.3. Mass Flow Split Boundary Conditions

In ANSYS FLUENT, it is possible to use multiple outflow boundaries and specify the fractional flow rate through each boundary. In the *Outflow Dialog Box* (p. 1836), set the **Flow Rate Weighting** to indicate what portion of the outflow is through the boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.23 The Outflow Dialog Box



The **Flow Rate Weighting** is a weighting factor:

$$\begin{matrix}\text{percentage flow}\\ \text{through boundary}\end{matrix} = \frac{\text{Flow Rate Weighting specified on boundary}}{\text{sum of all Flow Rate Weightings}} \tag{7–80}$$

By default, the **Flow Rate Weighting** for all outflow boundaries is set to 1. If the flow is divided equally among all of your outflow boundaries (or if you have just one outflow boundary), you need not change the settings from the default; ANSYS FLUENT will scale the flow rate fractions to obtain equal fractions through all outflow boundaries. Thus, if you have two outflow boundaries and you want half of the flow to exit through each one, no inputs are required from you. If, however, you want 75% of the flow to exit through one, and 25% through the other, you will need to explicitly specify *both* **Flow Rate Weighting** values, i.e., 0.75 for one boundary and 0.25 for the other.

Important

If you specify a **Flow Rate Weighting** of 0.75 at the first exit and leave the default **Flow Rate Weighting** (1.0) at the second exit, then the flow through each boundary will be

$$\text{Boundary 1} = \frac{0.75}{0.75+1.0} = 0.429 \text{ or } 42.9\%$$

$$\text{Boundary 2} = \frac{1.0}{0.75+1.0} = 0.571 \text{ or } 57.1\%$$

7.3.10.4. Other Inputs at Outflow Boundaries

7.3.10.4.1. Radiation Inputs at Outflow Boundaries

In general, there are no boundary conditions for you to set at an outflow boundary. If, however, you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature Method** in the **Outflow** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These parameters are described in *Defining Boundary Conditions for Radiation* (p. 724). The default value for **Internal Emissivity** is 1 and the default value for **Black Body Temperature** is 300.

7.3.10.4.2. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the outflow boundary. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.11. Outlet Vent Boundary Conditions

Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) pressure and temperature.

7.3.11.1. Inputs at Outlet Vent Boundaries

You will enter the following information for an outlet vent boundary:

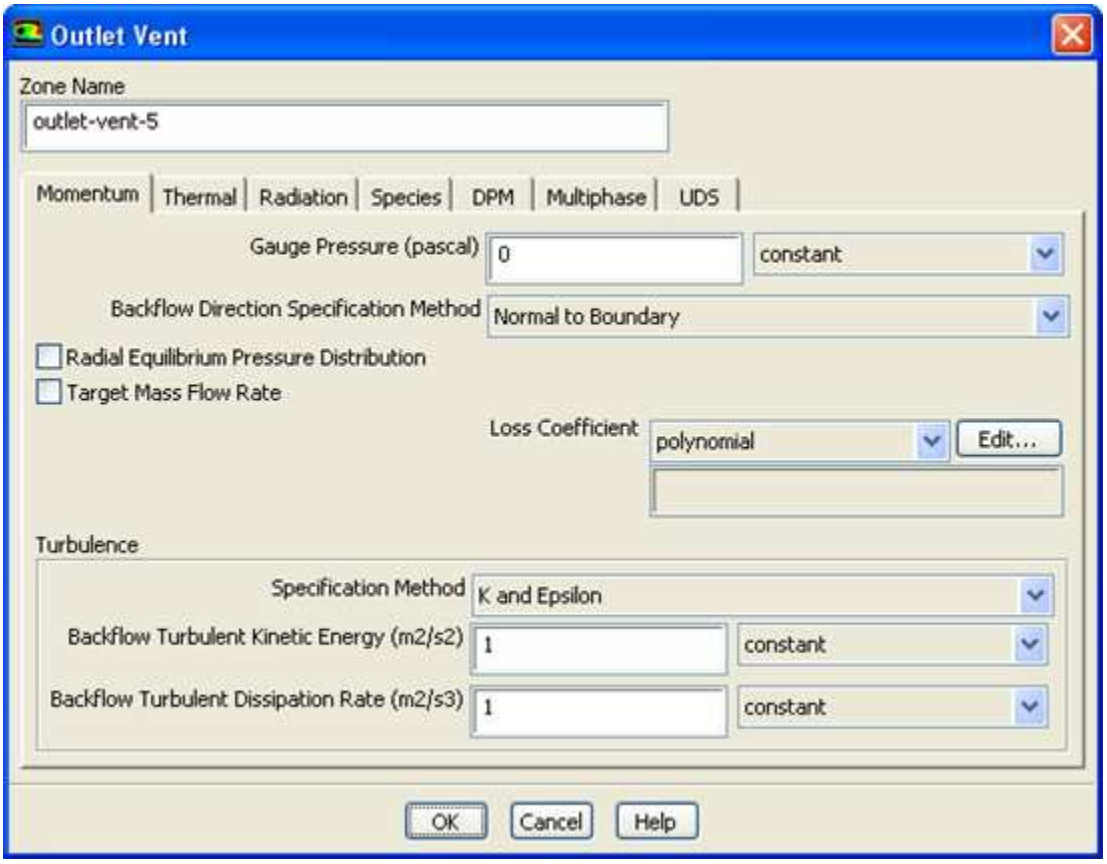
- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- discrete phase boundary conditions (for discrete phase calculations)
- loss coefficient
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the *Outlet Vent Dialog Box* (p. 1838) (*Figure 7.24* (p. 298)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See *Inputs at Pressure Outlet Boundaries* (p. 282) for details. Specification of the loss coefficient is described here. Open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.24 The Outlet Vent Dialog Box



7.3.11.1.1. Specifying the Loss Coefficient

An outlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient which you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the vent, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \qquad \textbf{(7–81)}$$

where ρ is the fluid density, and k_L is the nondimensional loss coefficient.

Important

Δp is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Loss Coefficient** across the vent. The dialog boxes for defining these functions are the sameas those used for defining temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.12. Exhaust Fan Boundary Conditions

Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) pressure.

7.3.12.1. Inputs at Exhaust Fan Boundaries

You will enter the following information for an exhaust fan boundary:

- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
  - user-defined scalar boundary conditions (for user-defined scalar calculations)
- radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- discrete phase boundary conditions (for discrete phase calculations)
- pressure jump
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the *Exhaust Fan Dialog Box* (p. 1815) (*Figure 7.25* (p. 300)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

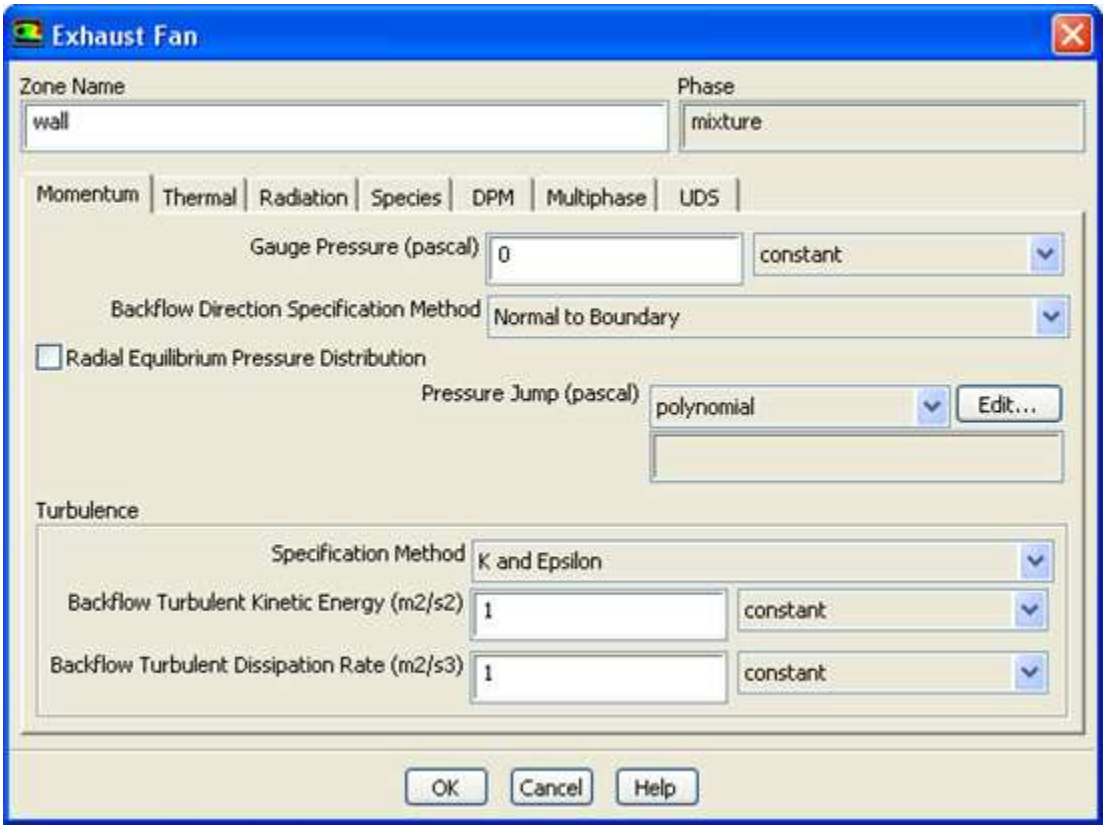
The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See *Inputs at Pressure Outlet Boundaries* (p. 282) for details. Specification of the pressure jump is described here. Open channel boundary condition inputs are described in *Modeling Open Channel Flows* (p. 1121).

7.3.12.1.1. Specifying the Pressure Jump

An exhaust fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the local fluid velocity normal to the fan. You can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Pressure Jump** across the fan. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.25 The Exhaust Fan Dialog Box



Important

You must be careful to model the exhaust fan so that a pressure rise occurs for forward flow through the fan. In the case of reversed flow, the fan is treated like an inlet vent with a loss coefficient of unity.

7.3.13. Wall Boundary Conditions

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear. (You can also model a slip wall with zero shear using the symmetry boundary type, but using a symmetry boundary will apply symmetry conditions for *all* equations. See *Symmetry Boundary Conditions* (p. 319) for details.)

The shear stress and heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

7.3.13.1. Inputs at Wall Boundaries

7.3.13.1.1. Summary

You will enter the following information for a wall boundary:

- wall motion conditions (for moving or rotating walls)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

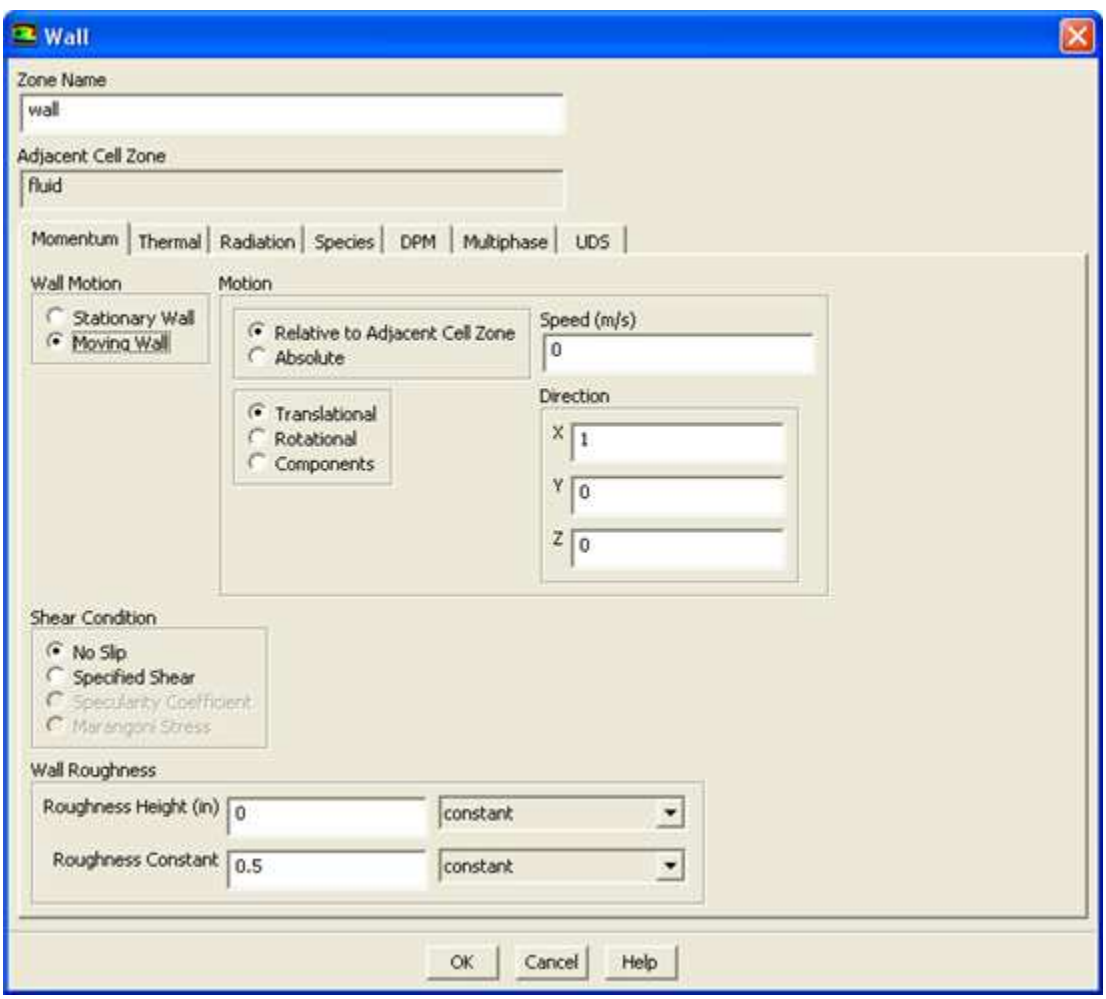
- shear conditions (for slip walls, optional)
- wall roughness (for turbulent flows, optional)
- thermal boundary conditions (for heat transfer calculations)
- species boundary conditions (for species calculations)
- chemical reaction boundary conditions (for surface reactions)
- radiation boundary conditions (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- discrete phase boundary conditions (for discrete phase calculations)
- wall adhesion contact angle (for VOF calculations, optional)

7.3.13.2. Wall Motion

Wall boundaries can be either stationary or moving. The stationary boundary condition specifies a fixed wall, whereas the moving boundary condition can be used to specify the translational or rotational velocity of the wall, or the velocity components.

Wall motion conditions are entered in the **Momentum** tab of the *Wall Dialog Box* (p. 1865) (*Figure 7.26* (p. 301)), which is opened from the *Boundary Conditions Task Page* (p. 1812) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)). To view the wall motion conditions, click the **Momentum** tab.

Figure 7.26 The Wall Dialog Box for a Moving Wall



7.3.13.2.1. Defining a Stationary Wall

For a stationary wall, choose the **Stationary Wall** option under **Wall Motion**.

7.3.13.2.2. Velocity Conditions for Moving Walls

If you wish to include tangential motion of the wall in your calculation, you need to define the translational or rotational velocity, or the velocity components. Select the **Moving Wall** option under **Wall Motion**. The **Wall** dialog box will expand, as shown in *Figure 7.26* (p. 301), to show the wall velocity conditions.

Note that you cannot use the moving wall condition to model problems where the wall has a motion normal to itself. ANSYS FLUENT will neglect any normal component of wall motion that you specify using the methods below.

Specifying Relative or Absolute Velocity

If the cell zone adjacent to the wall is moving (e.g., if you are using a moving reference frame or a sliding mesh), you can choose to specify velocities relative to the zone motion by enabling the **Relative to Adjacent Cell Zone** option. If you choose to specify relative velocities, a velocity of zero means that the wall is stationary in the relative frame, and therefore moving at the speed of the adjacent cell zone in the absolute frame. If you choose to specify absolute velocities (by enabling the **Absolute** option), a velocity of zero means that the wall is stationary in the absolute frame, and therefore moving at the speed of the adjacent cell zone—but in the opposite direction—in the relative reference frame.

Important

If you are using one or more moving reference frames, sliding meshes, or mixing planes, and you want the wall to be fixed in the moving frame, it is recommended that you specify relative velocities (the default) rather than absolute velocities. Then, if you modify the speed of the adjacent cell zone, you will not need to make any changes to the wall velocities, as you would if you specified absolute velocities.

Note that if the adjacent cell zone is not moving, the absolute and relative options are equivalent.

Translational Wall Motion

For problems that include linear translational motion of the wall boundary (e.g., a rectangular duct with a moving belt as one wall) you can enable the **Translational** option and specify the wall's **Speed** and **Direction** (**X,Y,Z** vector). By default, wall motion is "disabled" by the specification of **Translational** velocity with a **Speed** of zero.

If you need to define non-linear translational motion, you will need to use the **Components** option, described below.

Rotational Wall Motion

For problems that include rotational wall motion you can enable the **Rotational** option and define the rotational **Speed** about a specified axis. To define the axis, set the **Rotation-Axis Direction** and **Rotation-Axis Origin**. This axis is independent of the axis of rotation used by the adjacent cell zone, and independent of any other wall rotation axis. For 3D problems, the axis of rotation is the vector passing through the specified **Rotation-Axis Origin** and parallel to the vector from (0,0,0) to the (**X,Y,Z**) point specified under **Rotation-Axis Direction**. For 2D problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

z-direction vector passing through the specified point. For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

Note that the modeling of tangential rotational motion will be correct only if the wall bounds a surface of revolution about the prescribed axis of rotation (e.g., a circle or cylinder). Note also that rotational motion can be specified for a wall in a stationary reference frame.

Wall Motion Based on Velocity Components

For problems that include linear or non-linear translational motion of the wall boundary you can enable the **Components** option and specify the **X-Velocity**, **Y-Velocity**, and **Z-Velocity** of the wall. You can define non-linear translational motion using a profile or a user-defined function for the **X-Velocity**, **Y-Velocity**, and/or **Z-Velocity** of the wall.

Wall Motion for Two-Sided Walls

As discussed earlier in this section, when you read a mesh with a two-sided wall zone (which forms the interface between fluid/solid regions) into ANSYS FLUENT, a "shadow" zone will automatically be created so that each side of the wall is a distinct wall zone. For two-sided walls, it is possible to specify different motions for the wall and shadow zones, whether or not they are coupled. Note, however, that you cannot specify motion for a wall (or shadow) that is adjacent to a solid zone.

7.3.13.2.3. Shear Conditions at Walls

Four types of shear conditions are available:

- no-slip
- specified shear
- specularity coefficient
- Marangoni stress

The no-slip condition is the default, and it indicates that the fluid sticks to the wall and moves with the same velocity as the wall, if it is moving. The specified shear and Marangoni stress boundary conditions are useful in modeling situations in which the shear stress (rather than the motion of the fluid) is known. Examples of such situations are applied shear stress, slip wall (zero shear stress), and free surface conditions (zero shear stress or shear stress dependent on surface tension gradient). The specified shear boundary condition allows you to specify the x, y, and z components of the shear stress as constant values or profiles. The Marangoni stress boundary condition allows you to specify the gradient of the surface tension with respect to the temperature at this surface. The shear stress is calculated based on the surface gradient of the temperature and the specified surface tension gradient. The Marangoni stress option is available only for calculations in which the energy equation is being solved.

The specularity coefficient shear condition is specifically used in multiphase with granular flows. The specularity coefficient is a measure of the fraction of collisions which transfer momentum to the wall and its value ranges between zero and unity. This implementation is based on the Johnson and Jackson [*38*] (p. 2211) boundary conditions for granular flows.

Shear conditions are entered in the **Momentum** tab of the *Wall Dialog Box* (p. 1865), which is opened from the *Boundary Conditions Task Page* (p. 1812) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

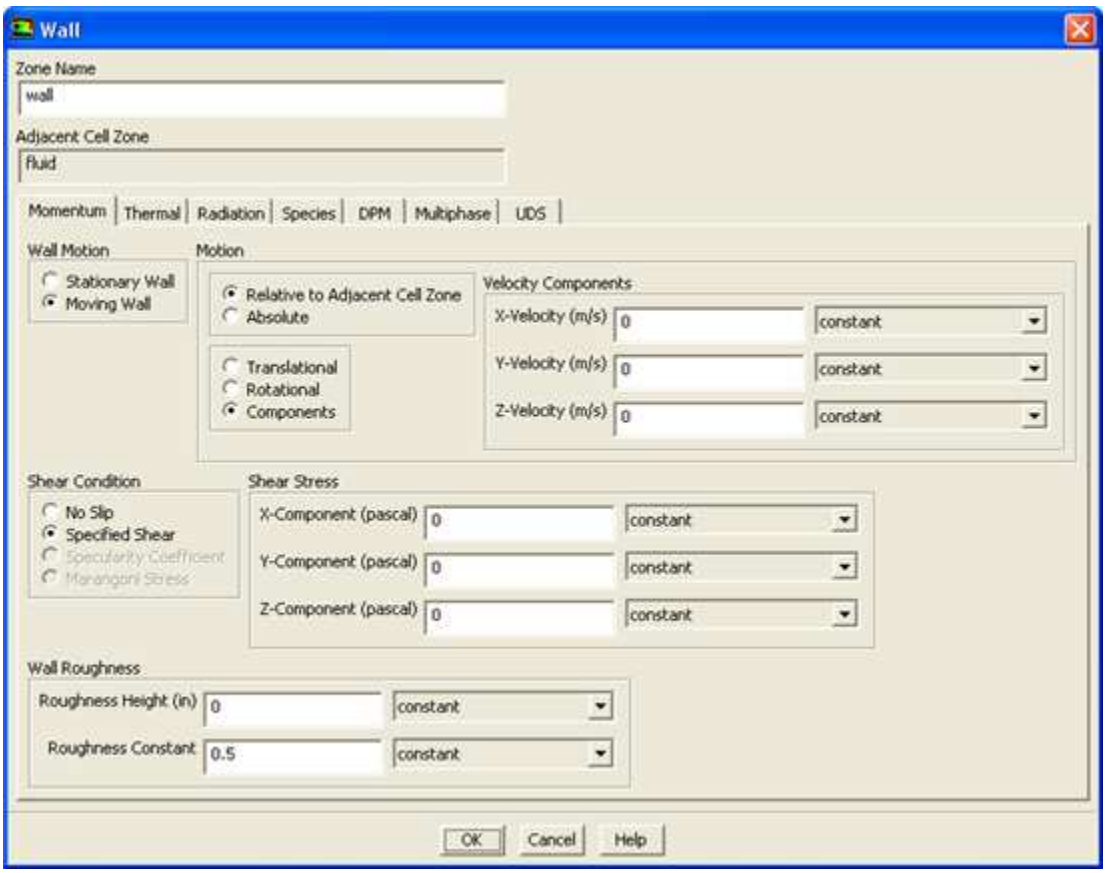
7.3.13.2.4. No-Slip Walls

You can model a no-slip wall by selecting the **No Slip** option under **Shear Condition**. This is the default for all walls in viscous flows.

7.3.13.2.5. Specified Shear

In addition to the no-slip wall that is the default for viscous flows, you can model a slip wall by specifying zero or non-zero shear. For non-zero shear, the shear to be specified is the shear at the wall by the fluid. To specify the shear, select the **Specified Shear** option under **Shear Condition** (see *Figure 7.27* (p. 304)). You can then enter x, y, and z components of shear under **Shear Stress**. Wall functions for turbulence are not used with the **Specified Shear** option.

Figure 7.27 The Wall Dialog Box for Specified Shear

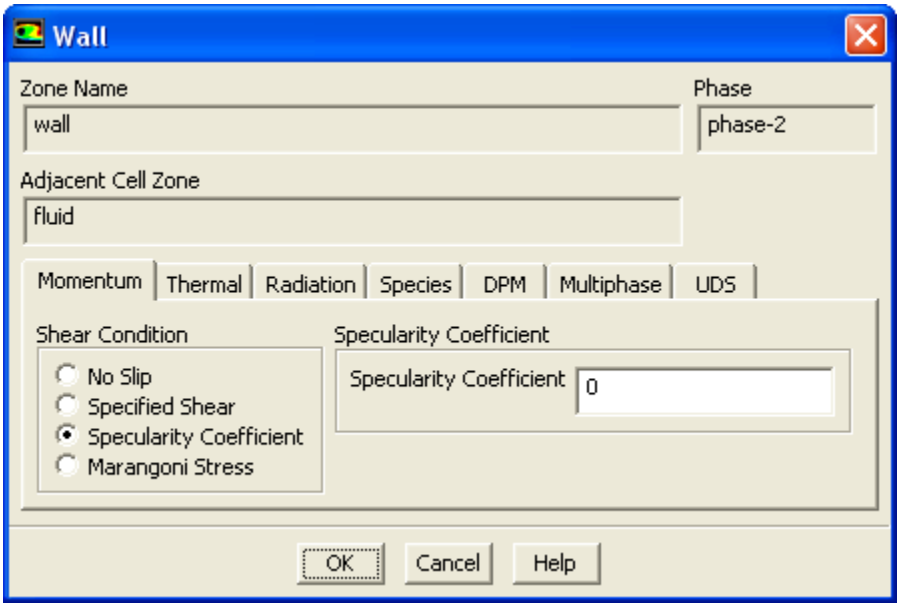


7.3.13.2.6. Specularity Coefficient

For multiphase granular flow, you can specify the specularity coefficient such that when the value is zero, this condition is equivalent to zero shear at the wall, but when the value is near unity, there is a significant amount of lateral momentum transfer. To specify the specularity coefficient, select the **Specularity Coefficient** option under **Shear Condition** (see *Figure 7.28* (p. 305)) and enter the desired value in the text-entry box under **Specularity Coefficient**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.28 The Wall Dialog Box for the Specularity Coefficient



7.3.13.2.7. Marangoni Stress

ANSYS FLUENT can also model shear stresses caused by the variation of surface tension due to temperature. The shear stress applied at the wall is given by

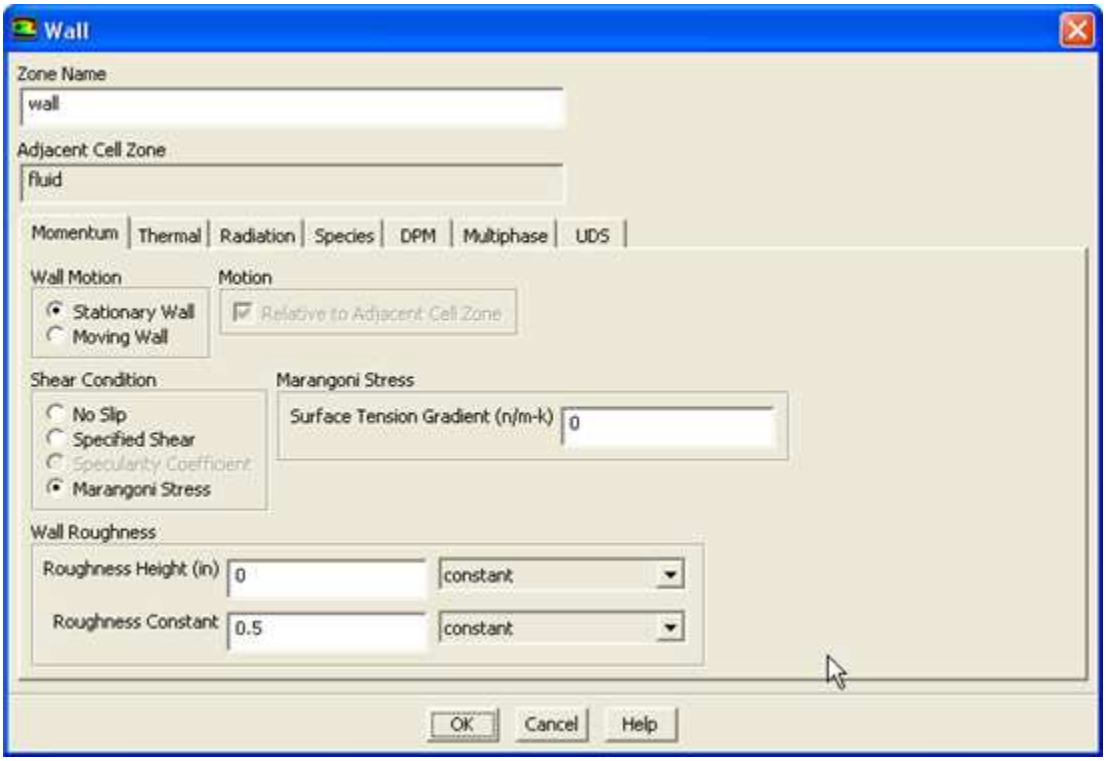
$$\tau = \frac{d\sigma}{dT} \nabla_s T \tag{7–82}$$

where $d\sigma/dT$ is the surface tension gradient with respect to temperature, and $\nabla_s T$ is the surface gradient. This shear stress is then applied to the momentum equation.

To model Marangoni stress for the wall, select the **Marangoni Stress** option under **Shear Condition** (see *Figure 7.29* (p. 306)). This option is available only for calculations in which the energy equation is being solved. You can then enter the surface tension gradient ($d\sigma/dT$ in *Equation 7–82* (p. 305)) in the **Surface Tension Gradient** field. Wall functions for turbulence are not used with the **Marangoni Stress** option.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.29 The Wall Dialog Box for Marangoni Stress



7.3.13.2.8. Wall Roughness Effects in Turbulent Wall-Bounded Flows

Fluid flows over rough surfaces are encountered in diverse situations. Examples are, among many others, flows over the surfaces of airplanes, ships, turbomachinery, heat exchangers, and piping systems, and atmospheric boundary layers over terrain of varying roughness. Wall roughness affects drag (resistance) and heat and mass transfer on the walls.

If you are modeling a turbulent wall-bounded flow in which the wall roughness effects are considered to be significant, you can include the wall roughness effects through the law-of-the-wall modified for roughness.

7.3.13.2.9. Law-of-the-Wall Modified for Roughness

Experiments in roughened pipes and channels indicate that the mean velocity distribution near rough walls, when plotted in the usual semi-logarithmic scale, has the same slope ($1/\kappa$) but a different intercept (additive constant B in the log-law). Thus, the law-of-the-wall for mean velocity modified for roughness has the form

$$\frac{u_p u^*}{\tau_w/\rho} = \frac{1}{\kappa}\ln\left(E\frac{\rho u^* y_p}{\mu}\right) - \Delta B \qquad \textbf{(7–83)}$$

where $u^* = C_\mu^{1/4} k^{1/2}$ and

$$\Delta B = \frac{1}{\kappa}\ln f_r \qquad \textbf{(7–84)}$$

where f_r is a roughness function that quantifies the shift of the intercept due to roughness effects.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ΔB depends, in general, on the type (uniform sand, rivets, threads, ribs, mesh-wire, etc.) and size of the roughness. There is no universal roughness function valid for all types of roughness. For a sand-grain roughness and similar types of uniform roughness elements, however, ΔB has been found to be well-correlated with the nondimensional roughness height, $K_s^+ = \rho K_s u^* / \mu$, where K_s is the physical roughness height and $u^* = C_\mu^{1/4} k^{1/2}$. Analyses of experimental data show that the roughness function is not a single function of K_s^+, but takes different forms depending on the K_s^+ value. It has been observed that there are three distinct regimes:

- hydrodynamically smooth ($K_s^+ \leq 2.25$)
- transitional ($2.25 < K_s^+ \leq 90$)
- fully rough ($K_s^+ > 90$)

According to the data, roughness effects are negligible in the hydrodynamically smooth regime, but become increasingly important in the transitional regime, and take full effect in the fully rough regime.

In ANSYS FLUENT, the whole roughness regime is subdivided into the three regimes, and the formulas proposed by Cebeci and Bradshaw based on Nikuradse's data [*14*] (p. 2209) are adopted to compute ΔB for each regime.

For the hydrodynamically smooth regime ($K_s^+ \leq 2.25$):

$$\Delta B = 0 \tag{7–85}$$

For the transitional regime ($2.25 < K_s^+ \leq 90$):

$$\Delta B = \frac{1}{\kappa} \ln\left[\frac{K_s^+ - 2.25}{87.75} + C_s K_s^+\right] \times \sin\{0.4258(\ln K_s^+ - 0.811)\} \tag{7–86}$$

where C_s is a roughness constant, and depends on the type of the roughness.

In the fully rough regime ($K_s^+ > 90$):

$$\Delta B = \frac{1}{\kappa} \ln(1 + C_s K_s^+) \tag{7–87}$$

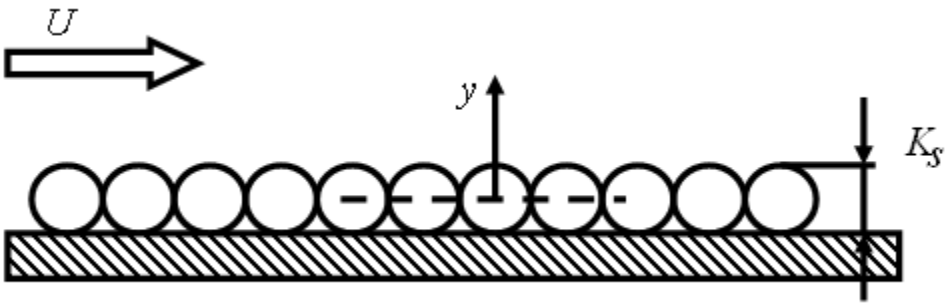
In the solver, given the roughness parameters, $\Delta B(K_s^+)$ is evaluated using the corresponding formula (*Equation 7–85* (p. 307), *Equation 7–86* (p. 307), or *Equation 7–87* (p. 307)). The modified law-of-the-wall in *Equation 7–83* (p. 306) is then used to evaluate the shear stress at the wall and other wall functions for the mean temperature and turbulent quantities.

The near wall treatment of smooth walls for turbulence models based on the ω-equation performs an optimized blending between the viscous sublayer and logarithmic region (see Wall Boundary Conditions and Enhanced Wall Treatment for Momentum and Energy Equations in the Theory Guide for more details). For rough walls, an additional modification has been made to account for the displacement caused by the surface roughness. The viscous sublayer is fully established only near hydraulically smooth walls. In the transitional

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

roughness regime, the roughness elements are slightly thicker than the viscous sublayer and start to disturb it, so that in fully rough flows, the sublayer is destroyed and viscous effects become negligible. The following figure illustrates the equivalent sand-grain roughness using a wall with a layer of closely packed spheres, which have an average roughness height (see for example, Schlichting and Gersten [*76*] (p. 2213)):

Figure 7.30 Illustration of Equivalent Sand-Grain Roughness



It can be assumed that the roughness has a blockage effect, which is about 50% of its height (note that the figure above shows a two-dimensional cut of a three-dimensional arrangement). This leads to the idea to shift the wall physically to 50% of the height of the roughness elements:

$$y^+ = y^+ + K_s^+ / 2 \qquad \textbf{(7–88)}$$

which gives about the correct displacement caused by the surface roughness. Due to this shift, the viscous sublayer formulation has only an influence for small values of K_s^+ in the improved rough wall treatment for turbulence models based on the ω-equation.

Please note that this shift is the default treatment for rough walls for turbulence models based on the ω-equation.

7.3.13.2.10. Setting the Roughness Parameters

The roughness parameters are in the **Momentum** tab of the *Wall Dialog Box* (p. 1865) (see *Figure 7.29* (p. 306)), which is opened from the *Boundary Conditions Task Page* (p. 1812) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

To model the wall roughness effects, you must specify two roughness parameters: the **Roughness Height**, K_s, and the **Roughness Constant**, C_s. The default roughness height (K_s) is zero, which corresponds to smooth walls. For the roughness to take effect, you must specify a non-zero value for K_s. For a uniform sand-grain roughness, the height of the sand-grain can simply be taken for K_s. For a non-uniform sand-grain, however, the mean diameter (D_{50}) would be a more meaningful roughness height. For other types of roughness, an "equivalent" sand-grain roughness height could be used for K_s. The above approaches are only relevant if the height is considered constant per surface. However, if the roughness constant or roughness height is not constant (i.e., flow over a nonuniform surface), then you can specify a profile (see *Profiles* (p. 366)). Similarly, user-defined functions may be used to define a wall roughness height that is not constant. For details on the format of user-defined functions, refer to the UDF Manual.

Choosing a proper roughness constant (C_s) is dictated mainly by the type of the given roughness. The default roughness constant ($C_s = 0.5$) was determined so that, when used with k- εturbulence models, it reproduces

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nikuradse's resistance data for pipes roughened with tightly-packed, uniform sand-grain roughness. You may need to adjust the roughness constant when the roughness you want to model departs much from uniform sand-grain. For instance, there is some experimental evidence that, for non-uniform sand-grains, ribs, and wire-mesh roughness, a higher value ($C_S = 0.5 \sim 1.0$) is more appropriate. Unfortunately, a clear guideline for choosing C_S for arbitrary types of roughness is not available.

Note that it is not physically meaningful to have a mesh size such that the wall-adjacent cell is smaller than the roughness height. For best results, make sure that the distance from the wall to the centroid of the wall-adjacent cell is greater than K_S.

7.3.13.3. Thermal Boundary Conditions at Walls

When you are solving the energy equation, you need to define thermal boundary conditions at wall boundaries. Five types of thermal conditions are available:

- fixed heat flux
- fixed temperature
- convective heat transfer
- external radiation heat transfer
- combined external radiation and convection heat transfer

If the wall zone is a "two-sided wall" (a wall that forms the interface between two regions, such as the fluid/solid interface for a conjugate heat transfer problem) a subset of these thermal conditions will be available, but you will also be able to choose whether or not the two sides of the wall are "coupled". See below for details.

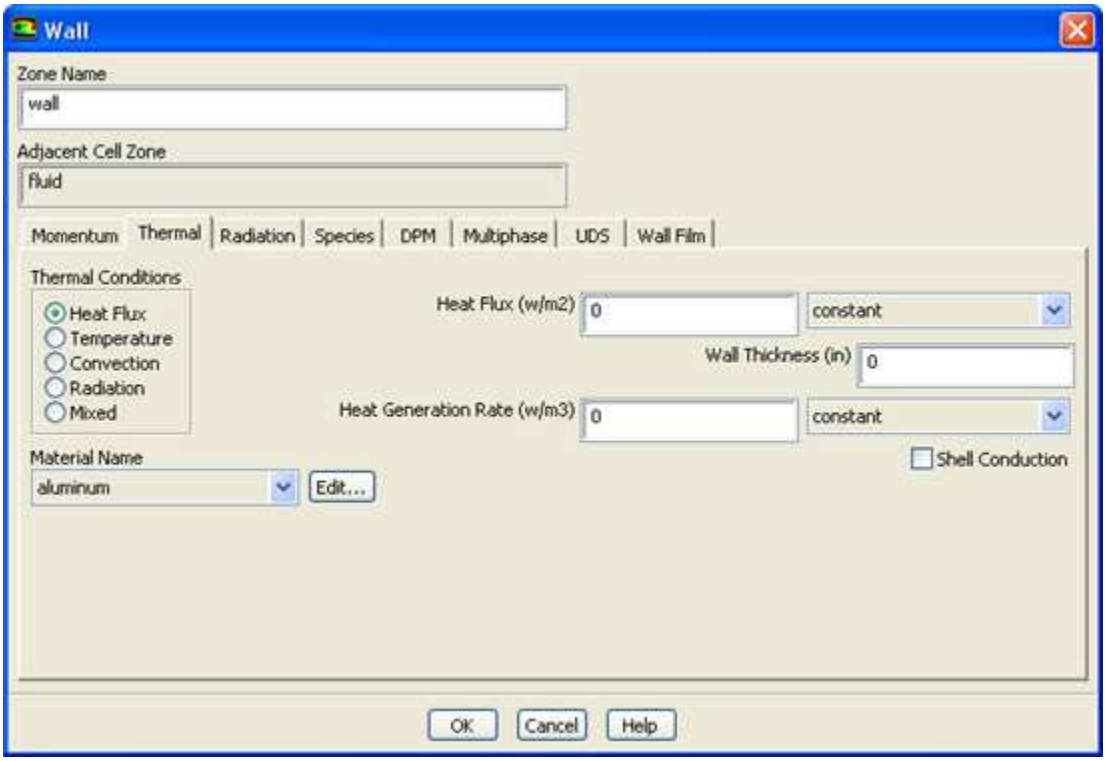
The inputs for each type of thermal condition are described below. If the wall has a non-zero thickness, you should also set parameters for calculating thin-wall thermal resistance and heat generation in the wall, as described below.

You can model conduction within boundary walls and internal (i.e., two-sided) walls of your model. This type of conduction, called shell conduction, allows you to more conveniently model heat conduction on walls where the wall thickness is small with respect to the overall geometry (e.g., finned heat exchangers or sheet metal in automobile underhoods). Meshing these walls with solid cells would lead to high-aspect-ratio meshes and a significant increase in the total number of cells. See below for details about shell conduction.

Thermal conditions are entered in the **Thermal** tab of the *Wall Dialog Box* (p. 1865) (*Figure 7.31* (p. 310)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.31 The Wall Dialog Box (Thermal Tab)



7.3.13.3.1. Heat Flux Boundary Conditions

For a fixed heat flux condition, choose the **Heat Flux** option under **Thermal Conditions**. You will then need to set the appropriate value for the heat flux at the wall surface in the **Heat Flux** field. You can define an adiabatic wall by setting a zero heat flux condition. This is the default condition for all walls.

7.3.13.3.2. Temperature Boundary Conditions

To select the fixed temperature condition, choose the **Temperature** option under **Thermal Conditions** in the **Wall** dialog box. You will need to specify the temperature at the wall surface (**Temperature**). The heat transfer to the wall is computed using *Equation 7–90* (p. 317) or *Equation 7–91* (p. 317).

7.3.13.3.3. Convective Heat Transfer Boundary Conditions

For a convective heat transfer wall boundary, select **Convection** under **Thermal Conditions**. Your inputs of **Heat Transfer Coefficient** and **Free Stream Temperature** will allow ANSYS FLUENT to compute the heat transfer to the wall using *Equation 7–94* (p. 318).

7.3.13.3.4. External Radiation Boundary Conditions

If radiation heat transfer from the exterior of your model is of interest, you can enable the **Radiation** option in the **Wall** dialog box and set the **External Emissivity** and **External Radiation Temperature**.

7.3.13.3.5. Combined Convection and External Radiation Boundary Conditions

You can choose a thermal condition that combines the convection and radiation boundary conditions by selecting the **Mixed** option. With this thermal condition, you will need to set the **Heat Transfer Coefficient**, **Free Stream Temperature**, **External Emissivity**, and **External Radiation Temperature**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.13.3.6. Thin-Wall Thermal Resistance Parameters

By default, a wall will have a thickness of zero. You can, however, in conjunction with any of the thermal conditions, model a thin layer of material on the wall. For example, you can model the effect of a piece of sheet metal between two fluid zones, a coating on a solid zone, or contact resistance between two solid regions. ANSYS FLUENT will solve a 1D steady heat conduction equation to compute the thermal resistance offered by the wall and the heat generation in the wall.

To include these effects in the heat transfer calculation you will need to specify the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the **Material Name** drop-down list, and specify the thickness in the **Wall Thickness** field. If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

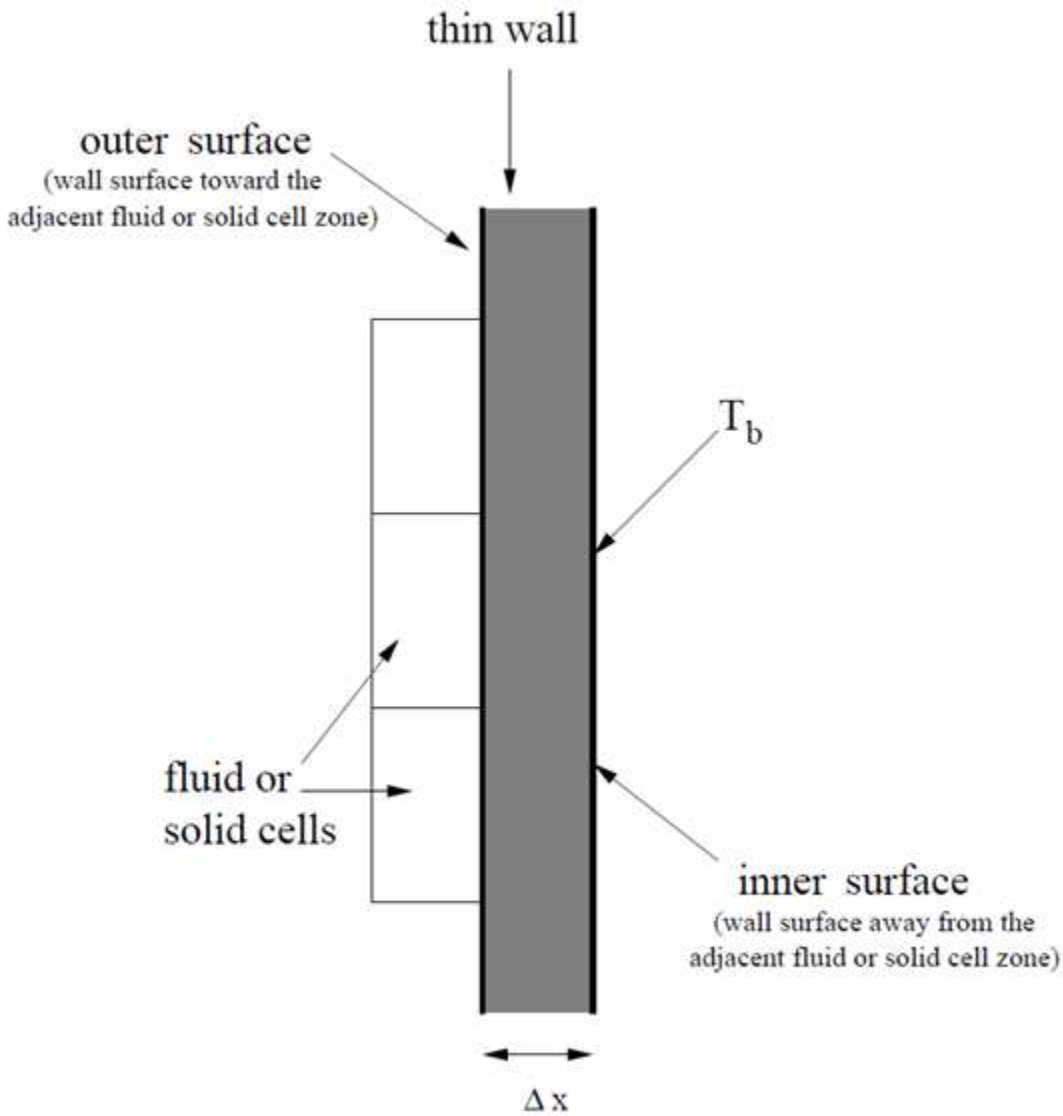
The thermal resistance of the wall is $\Delta x/k$, where k is the conductivity of the wall material and Δx is the wall thickness. The thermal wall boundary condition you set will be specified on the outside of the fluid/solid domain, which is called the inner surface of the thin wall, as shown in *Figure 7.32* (p. 312). This is the side of the wall surface away from the adjacent fluid or solid cell zone. The temperature specified at this side of the wall is T_b.

Important

The convention used in ANSYS FLUENT is that for any wall, "outer" refers to the surface of the wall facing the fluid/solid cell zone and "inner" refers to the surface of the wall facing away from the adjacent fluid/solid cell zone. If shell conduction is enabled (*Shell Conduction in Thin-Walls* (p. 313)), the shell cell temperature will be stored in the "inner" surface. If there is no shell conduction, then the "outer" surface stores the face temperature of the wall while the "inner" surface stores the evaluated value of the boundary condition specified by the user.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.32 Thermal Conditions are Specified on the Inner Surface of the Thin Wall



Important

Note that for thin walls, you can only specify a constant thermal conductivity. If you want to use a non-constant thermal conductivity for a wall with non-zero thickness, you should use the shell conduction model (see below for details).

Specify the heat generation rate inside the wall in the **Heat Generation Rate** field. This option is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits.

7.3.13.3.7. Thermal Conditions for Two-Sided Walls

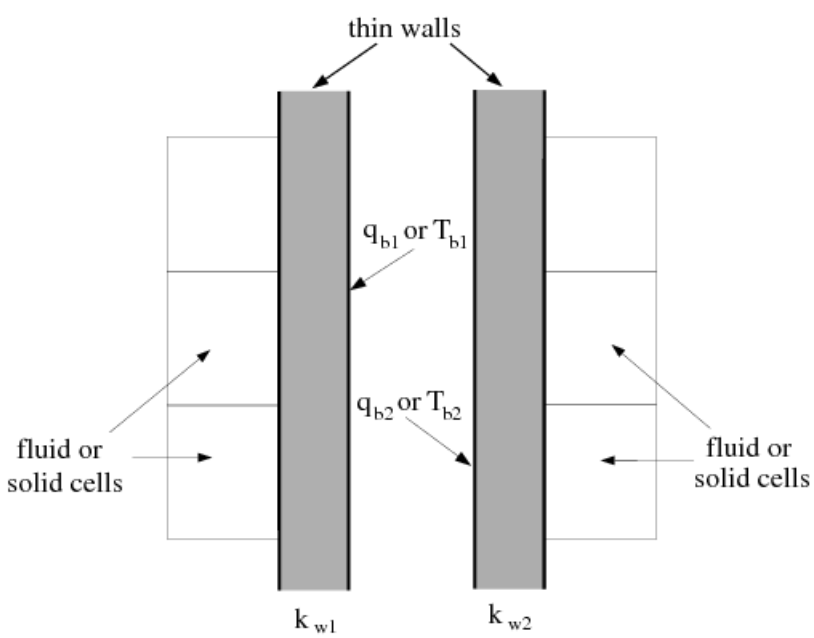
If the wall zone has a fluid or solid region on each side, it is called a "two-sided wall". When you read a mesh with this type of wall zone into ANSYS FLUENT, a "shadow" zone will automatically be created so that each side of the wall is a distinct wall zone. In the **Wall** dialog box, the shadow zone's name will be shown in the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shadow Face Zone field. You can choose to specify different thermal conditions on each zone, or to couple the two zones:

- To couple the two sides of the wall, select the **Coupled** option under **Thermal Conditions**. (This option will appear in the **Wall** dialog box only when the wall is a two-sided wall.) No additional thermal boundary conditions are required, because the solver will calculate heat transfer directly from the solution in the adjacent cells. You can, however, specify the material type, wall thickness, and heat generation rate for thin-wall thermal resistance calculations, as described above. Note that the resistance parameters you set for one side of the wall will automatically be assigned to its shadow wall zone. Specifying the heat generation rate inside the wall is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits but not the heat flux or wall temperature.
- To uncouple the two sides of the wall and specify different thermal conditions on each one, choose **Temperature** or **Heat Flux** as the thermal condition type. (**Convection** and **Radiation** are not applicable for two-sided walls.) The relationship between the wall and its shadow will be retained, so that you can couple them again at a later time, if desired. You will need to set the relevant parameters for the selected thermal condition, as described above. The two uncoupled walls can have different thicknesses, and are effectively insulated from one another. If you specify a non-zero wall thickness for the uncoupled walls, the thermal boundary conditions you set will be specified on the inner surfaces of the two thin walls, as shown in *Figure 7.33* (p. 313), where T_{b1} is the **Temperature** (or q_{b1} is the **Heat Flux**) specified on one wall and T_{b2} is the **Temperature** (or q_{b2} is the **Heat Flux**) specified on the other wall. k_{w1} and k_{w2} are the thermal conductivities of the uncoupled thin walls. Note that the gap between the walls in *Figure 7.33* (p. 313) is not part of the model; it is included in the figure only to show where the thermal boundary condition for each uncoupled wall is applied.

Figure 7.33 Thermal Conditions are Specified on the Inner Surfaces of the Uncoupled Thin Walls



7.3.13.3.8. Shell Conduction in Thin-Walls

To enable shell conduction for a wall, turn on the **Shell Conduction** option in the **Wall** boundary condition dialog box. When this option is enabled, ANSYS FLUENT will compute heat conduction within the wall, in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

addition to conduction across the wall (which is always computed when the energy equation is solved). The **Shell Conduction** option will appear in the **Wall** dialog box for all walls when solution of the energy equation is active. For a wall with shell conduction enabled, the thermal conditions are applied as described above for thin walls.

ANSYS FLUENT cases with shell conduction can be read in serial or parallel. Either a partitioned or an unpartitioned case file can be read in parallel (see *Mesh Partitioning and Load Balancing* (p. 1603) for more information on partitioning). After reading a case file in parallel, shell zones can be created on any wall with a positive thickness.

To delete existing shell conduction zones all at once, the TUI command `define/boundary-conditions/modify-zones/delete-all-shells` is used. This capability is available in both serial and parallel mode.

Important

You must specify a non-zero **Wall Thickness** in the **Wall** dialog box, because the shell conduction model is relevant only for walls with non-zero thickness.

Important

Note that the shell conduction model has several limitations:

- It is available only in 3D.
- It is available only when the pressure-based solver is used.
- It cannot be used with the non-premixed or partially premixed combustion model.
- Shell conducting walls cannot be split or merged. If you need to split or merge a shell conducting wall, disable the **Shell Conduction** option for the wall, perform the split or merge operation, and then enable **Shell Conduction** for the new wall zones.
- The shell conduction model cannot be used on a wall zone that has been adapted. If you want to perform adaption elsewhere in the computational domain, be sure to use the mask register described in *Manipulating Adaption Registers* (p. 1343)
- Fluxes at the ends of a shell conducting wall are not included in the heat balance reports. These fluxes are accounted for correctly in the ANSYS FLUENT solution, but not in the flux report itself.

7.3.13.4. Species Boundary Conditions for Walls

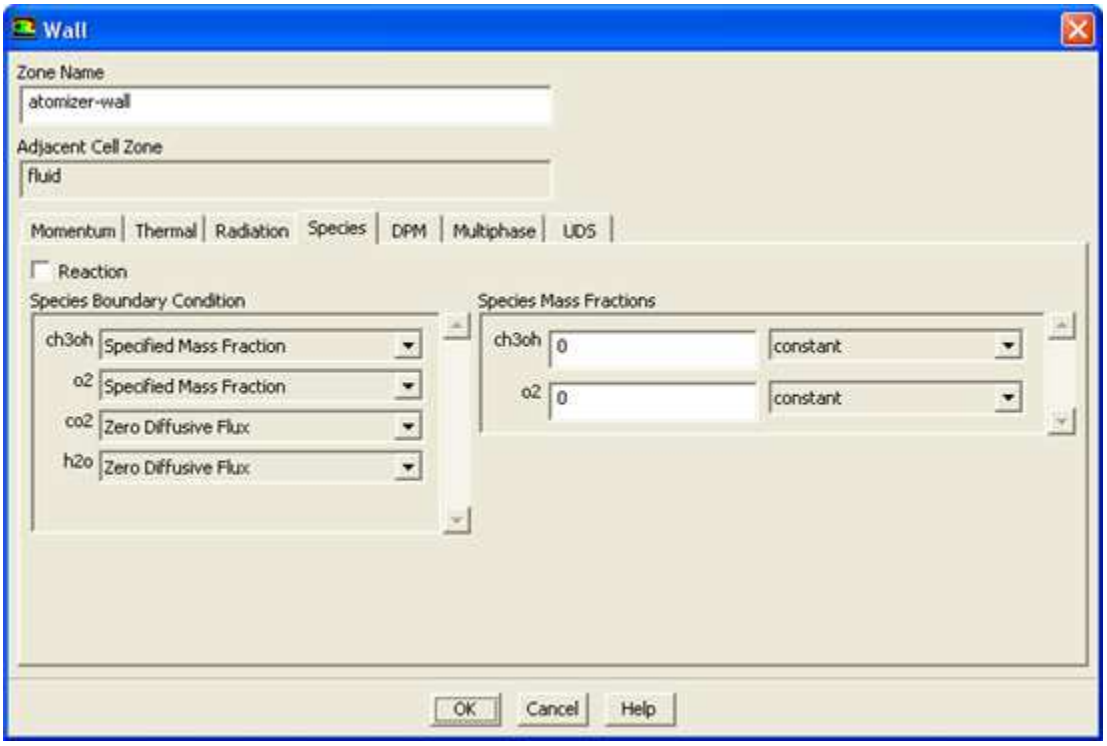
By default, a zero-gradient condition for all species is assumed at walls (except for species that participate in surface reactions), but it is also possible to specify species mass fractions at walls. That is, Dirichlet boundary conditions such as those that are specified at inlets can be used at walls as well.

If you wish to retain the default zero-gradient condition for a species, no inputs are required. If you want to specify the mass fraction for a species at the wall, the steps are as follows:

1. Click the **Species** tab in the *Wall Dialog Box* (p. 1865) to view the species boundary conditions for the wall (see *Figure 7.34* (p. 315)).
2. Under **Species Boundary Condition**, select **Specified Mass Fraction** (rather than **Zero Diffusive Flux**) in the drop-down list to the right of the species name. The dialog box will expand to include a field for **Species Mass Fractions**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.34 The Wall Dialog Box for Species Boundary Condition Input



3. Under **Species Mass Fractions**, specify the mass fraction for the species.

The boundary condition type for each species is specified separately, so you can choose to use different methods for different species.

If you are modeling species transport with reactions, you can, alternatively, enable a reaction mechanism at a wall by turning on the **Reaction** option and selecting an available mechanism from the **Reaction Mechanisms** drop-down list. See *Defining Zone-Based Reaction Mechanisms* (p. 816) for more information about defining reaction mechanisms.

You can also model unresolved surface washcoats, which greatly increase the catalytic surface area, by specifying the **Surface Area Washcoat Factor**. The surface washcoat increases the area available for surface reaction.

7.3.13.4.1. Reaction Boundary Conditions for Walls

If you have enabled the modeling of wall surface reactions in the *Species Model Dialog Box* (p. 1681), you can indicate whether or not surface reactions should be activated for the wall. In the **Species** tab of the **Wall** dialog box (*Figure 7.34* (p. 315)), turn the **Surface Reactions** option on or off.

Note that a zero-gradient condition is assumed at the wall for species that do not participate in any surface reactions.

7.3.13.5. Radiation Boundary Conditions for Walls

If you are using the gray P-1 radiation model, the DTRM, the gray DO model, or the surface-to-surface model, you will need to set the emissivity of the wall (**Internal Emissivity**) in the **Thermal** tab of the **Wall** dialog box. If you are using the Rosseland model you do not need to set the emissivity, because ANSYS FLUENT assumes the emissivity is 1.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the non-gray P–1 and the non-gray DO model, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box (the default value in each band is 1). Alternatively, you can specify the internal emissivity using a boundary condition parameter (see *Creating a New Parameter* (p. 211)). If you are using the non-gray DO model, you will also need to define the wall as opaque or semi-transparent in the **Radiation** tab. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

7.3.13.6. Discrete Phase Model (DPM) Boundary Conditions for Walls

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the wall in the **DPM** section of the **Wall** dialog box. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.13.6.1. Wall Adhesion Contact Angle for VOF Model

If you are using the VOF model and you are modeling wall adhesion, you can specify the contact angle for each pair of phases at the wall in the **Momentum** tab of the **Wall** dialog box. See *Steps for Setting Boundary Conditions* (p. 1115) for details.

7.3.13.7. User-Defined Scalar (UDS) Boundary Conditions for Walls

If you have defined UDS transport equations in your model, you can specify boundary conditions for each equation in the **UDS** section of the **Wall** dialog box. See *Setting Up UDS Equations in ANSYS FLUENT* (p. 488) for details.

7.3.13.8. Default Settings at Wall Boundaries

The default thermal boundary condition is a fixed heat flux of zero. Walls are, by default, not moving.

7.3.13.9. Shear-Stress Calculation Procedure at Wall Boundaries

For no-slip wall conditions, ANSYS FLUENT uses the properties of the flow adjacent to the wall/fluid boundary to predict the shear stress on the fluid at the wall. In laminar flows this calculation simply depends on the velocity gradient at the wall, while in turbulent flows one of the approaches described in Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide is used.

For specified-shear walls, ANSYS FLUENT will compute the tangential velocity at the boundary.

If you are modeling inviscid flow with ANSYS FLUENT, all walls use a slip condition, so they are frictionless and exert no shear stress on the adjacent fluid.

7.3.13.9.1. Shear-Stress Calculation in Laminar Flow

In a laminar flow, the wall shear stress is defined by the normal velocity gradient at the wall as

$$\tau_w = \mu \frac{\partial v}{\partial n} \tag{7–89}$$

When there is a steep velocity gradient at the wall, you must be sure that the mesh is sufficiently fine to accurately resolve the boundary layer. Guidelines for the appropriate placement of the near-wall node in laminar flows are provided in *Mesh Element Distribution* (p. 146).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.13.9.2. Shear-Stress Calculation in Turbulent Flows

Wall treatments for turbulent flows are described in Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide.

7.3.13.10. Heat Transfer Calculations at Wall Boundaries

7.3.13.10.1. Temperature Boundary Conditions

When a fixed temperature condition is applied at the wall, the heat flux to the wall from a fluid cell is computed as

$$q = h_f \left(T_w - T_f \right) + q_{rad} \tag{7–90}$$

where

h_f = fluid-side local heat transfer coefficient

T_w = wall surface temperature

T_f = local fluid temperature

q_{rad} = radiative heat flux

Note that the fluid-side heat transfer coefficient is computed based on the local flow-field conditions (e.g., turbulence level, temperature, and velocity profiles), as described by *Equation 7–97* (p. 319).

Heat transfer to the wall boundary from a solid cell is computed as

$$q = \frac{k_s}{\Delta n} (T_w - T_s) + q_{rad} \tag{7–91}$$

where

k_s = thermal conductivity of the solid

T_s = local solid temperature

Δn = distance between wall surface and the solid cell center

7.3.13.10.2. Heat Flux Boundary Conditions

When you define a heat flux boundary condition at a wall, you specify the heat flux at the wall surface. ANSYS FLUENT uses *Equation 7–90* (p. 317) and your input of heat flux to determine the wall surface temperature adjacent to a fluid cell as

$$T_w = \frac{q - q_{rad}}{h_f} + T_f \tag{7–92}$$

where, as noted above, the fluid-side heat transfer coefficient is computed based on the local flow-field conditions. When the wall borders a solid region, the wall surface temperature is computed as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T_w = \frac{(q - q_{rad})\Delta n}{k_s} + T_s \tag{7–93}$$

7.3.13.10.3. Convective Heat Transfer Boundary Conditions

When you specify a convective heat transfer coefficient boundary condition at a wall, ANSYS FLUENT uses your inputs of the external heat transfer coefficient and external heat sink temperature to compute the heat flux to the wall as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= h_{ext}(T_{ext} - T_w) \end{aligned} \tag{7–94}$$

where

h_{ext} = external heat transfer coefficient defined by you

T_{ext} = external heat-sink temperature defined by you

q_{rad} = radiative heat flux

Equation 7–94 (p. 318) assumes a wall of zero thickness.

7.3.13.10.4. External Radiation Boundary Conditions

When the external radiation boundary condition is used in ANSYS FLUENT, the heat flux to the wall is computed as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= \varepsilon_{ext}\sigma\left(T_\infty^4 - T_w^4\right) \end{aligned} \tag{7–95}$$

where

ε_{ext} = emissivity of the external wall surface defined by you

σ = Stefan-Boltzmann constant

T_w = surface temperature of the wall

T_∞ = temperature of the radiation source or sink on the exterior of the domain, defined by you

q_{rad} = radiative heat flux to the wall from within the domain

Equation 7–95 (p. 318) assumes a wall of zero thickness.

7.3.13.10.5. Combined External Convection and Radiation Boundary Conditions

When you choose the combined external heat transfer condition, the heat flux to the wall is computed as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$
\begin{aligned}
q &= h_f\left(T_w - T_f\right) + q_{rad} \\
&= h_{ext}\left(T_{ext} - T_w\right) + \varepsilon_{ext}\sigma\left(T_\infty^4 - T_w^4\right)
\end{aligned}
\tag{7–96}
$$

where the variables are as defined above. *Equation 7–96* (p. 319) assumes a wall of zero thickness.

7.3.13.10.6. Calculation of the Fluid-Side Heat Transfer Coefficient

In laminar flows, the fluid side heat transfer at walls is computed using Fourier's law applied at the walls. ANSYS FLUENT uses its discrete form:

$$
q = k_f\left(\frac{\partial T}{\partial n}\right)_{wall} \tag{7–97}
$$

where n is the local coordinate normal to the wall.

For turbulent flows, ANSYS FLUENT uses the law-of-the-wall for temperature derived using the analogy between heat and momentum transfer [*43*] (p. 2211). See Standard Wall Functions in the separate Theory Guide for details.

7.3.14. Symmetry Boundary Conditions

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow/thermal solution, have mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows. This section describes the treatment of the flow at symmetry planes and provides examples of the use of symmetry. You do not define any boundary conditions at symmetry boundaries, but you must take care to correctly define your symmetry boundary locations.

Important

At the centerline of an axisymmetric geometry, you should use the axis boundary type rather than the symmetry boundary type, as illustrated in *Figure 7.42* (p. 323). See *Axis Boundary Conditions* (p. 323) for details.

7.3.14.1. Examples of Symmetry Boundaries

Symmetry boundaries are used to reduce the extent of your computational model to a symmetric subsection of the overall physical system. *Figure 7.35* (p. 320) and *Figure 7.36* (p. 320) illustrate two examples of symmetry boundary conditions used in this way.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.35 Use of Symmetry to Model One Quarter of a 3D Duct

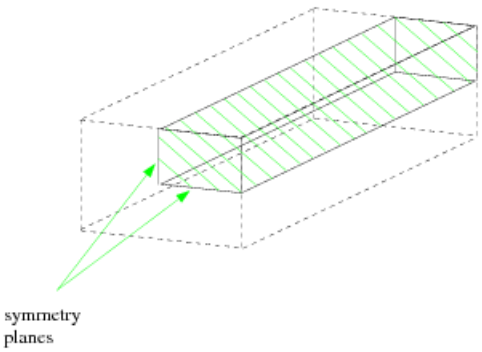


Figure 7.36 Use of Symmetry to Model One Quarter of a Circular Cross-Section

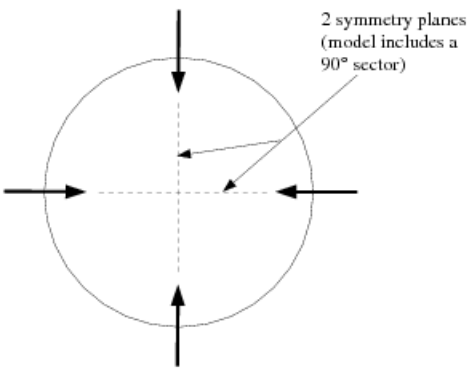
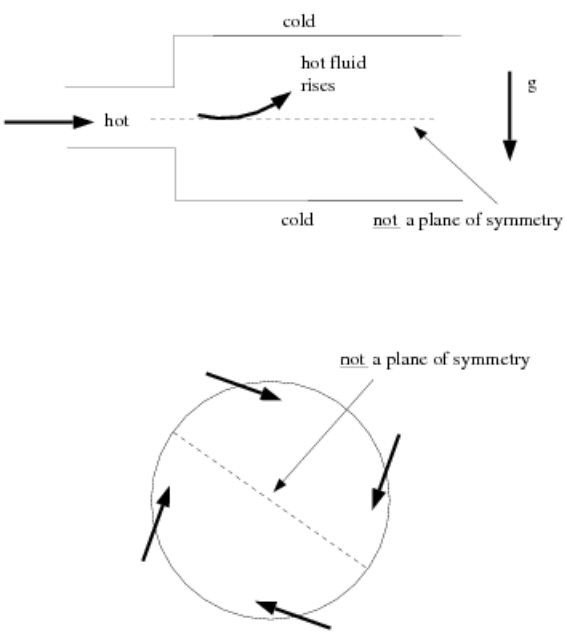


Figure 7.37 (p. 320) illustrates two problems in which a symmetry plane would be *inappropriate*. In both examples, the problem geometry is symmetric but the flow itself does not obey the symmetry boundary conditions. In the first example, buoyancy creates an asymmetric flow. In the second, swirl in the flow creates a flow normal to the would-be symmetry plane. Note that this second example should be handled using rotationally periodic boundaries (as illustrated in *Figure 7.38* (p. 321)).

Figure 7.37 Inappropriate Use of Symmetry



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.14.2. Calculation Procedure at Symmetry Boundaries

ANSYS FLUENT assumes a zero flux of all quantities across a symmetry boundary. There is no convective flux across a symmetry plane: the normal velocity component at the symmetry plane is thus zero. There is no diffusion flux across a symmetry plane: the normal gradients of all flow variables are thus zero at the symmetry plane. The symmetry boundary condition can therefore be summarized as follows:

- zero normal velocity at a symmetry plane
- zero normal gradients of all variables at a symmetry plane

As stated above, these conditions determine a zero flux across the symmetry plane, which is required by the definition of symmetry. Since the shear stress is zero at a symmetry boundary, it can also be interpreted as a "slip" wall when used in viscous flow calculations.

7.3.15. Periodic Boundary Conditions

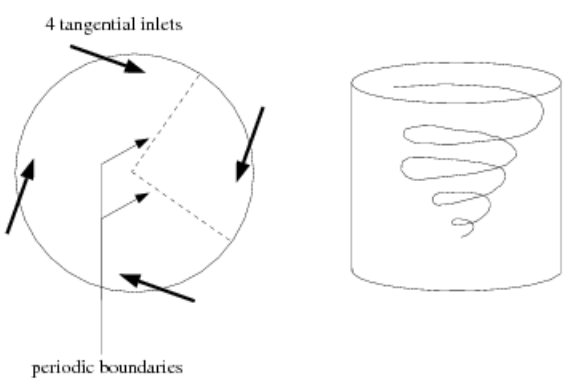
Periodic boundary conditions are used when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic conditions are available in ANSYS FLUENT. The first type does not allow a pressure drop across the periodic planes. (Note to FLUENT 4 users: This type of periodic boundary is referred to as a "cyclic" boundary in FLUENT 4.) The second type allows a pressure drop to occur across translationally periodic boundaries, enabling you to model "fully-developed" periodic flow. (In FLUENT 4 this is a "periodic" boundary.)

This section discusses the no-pressure-drop periodic boundary condition. A complete description of the fully-developed periodic flow modeling capability is provided in *Periodic Flows* (p. 493).

7.3.15.1. Examples of Periodic Boundaries

Periodic boundary conditions are used when the flows across two opposite planes in your computational model are identical. *Figure 7.38* (p. 321) illustrates a typical application of periodic boundary conditions. In this example the flow entering the computational model through one periodic plane is identical to the flow exiting the domain through the opposite periodic plane. Periodic planes are always used in pairs as illustrated in this example.

Figure 7.38 Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel



7.3.15.2. Inputs for Periodic Boundaries

For a periodic boundary without any pressure drop, there is only one input you need to consider: whether the geometry is rotationally or translationally periodic. (Additional inputs are required for a periodic flow with a periodic pressure drop. See *Periodic Flows* (p. 493).)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotationally periodic boundaries are boundaries that form an included angle about the centerline of a rotationally symmetric geometry. *Figure 7.38* (p. 321) illustrates rotational periodicity. Translationally periodic boundaries are boundaries that form periodic planes in a rectilinear geometry. *Figure 7.39* (p. 322) illustrates translationally periodic boundaries.

Figure 7.39 Example of Translational Periodicity - Physical Domain

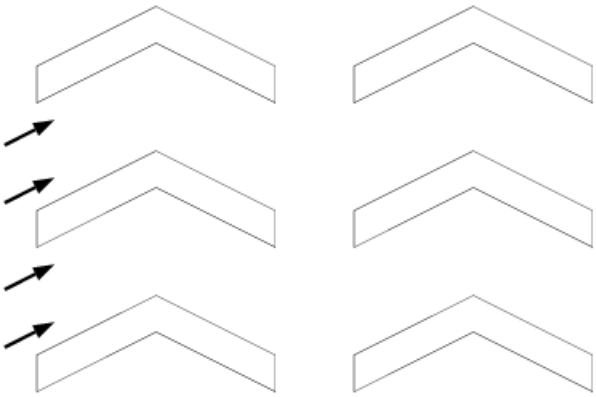
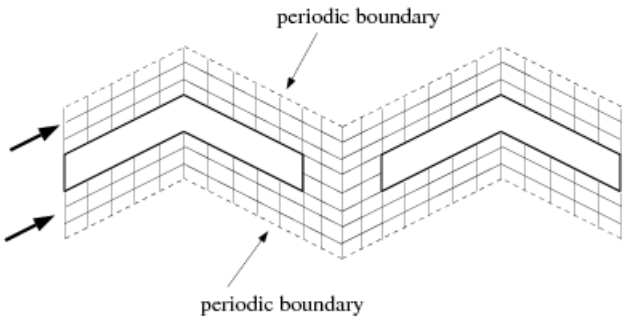
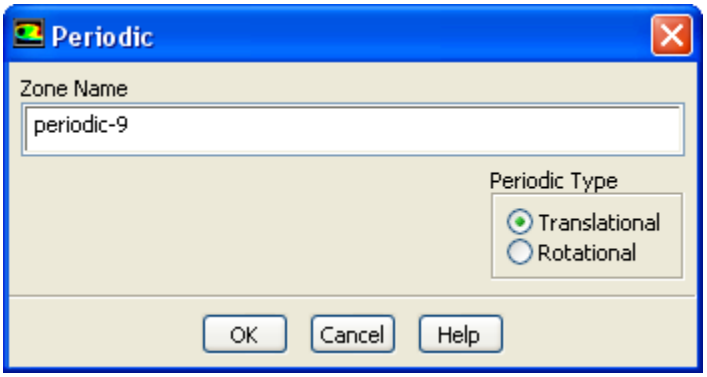


Figure 7.40 Example of Translational Periodicity - Modeled Domain



You will specify translational or rotational periodicity for a periodic boundary in the *Periodic Dialog Box* (p. 1842) (*Figure 7.41* (p. 322)), which is opened from the **Boundary Conditions** task page (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Figure 7.41 The Periodic Dialog Box



Note that there will be an additional item in the **Periodic** dialog box for the density-based solver, which allows you to specify the periodic pressure jump. See *Periodic Flows* (p. 493) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the domain is rotationally periodic, select **Rotational** as the **Periodic Type**; if it is translationally periodic, select **Translational**. For rotationally periodic domains, the solver will automatically compute the angle through which the periodic zone is rotated. The axis used for this rotation is the axis of rotation specified for the adjacent cell zone.

Note that there is no need for the adjacent cell zone to be moving for you to use a rotationally periodic boundary. You could, for example, model pipe flow in 3D using a stationary reference frame with a pie-slice of the pipe; the sides of the slice would require rotational periodicity.

You can use the **Mesh/Check** menu item (see *Checking the Mesh* (p. 172)) to compute and display the minimum, maximum, and average rotational angles of all faces on periodic boundaries. If the difference between the minimum, maximum, and average values is not negligible, then there is a problem with the mesh: the mesh geometry is not periodic about the specified axis.

7.3.15.3. Default Settings at Periodic Boundaries

By default, all periodic boundaries are translational.

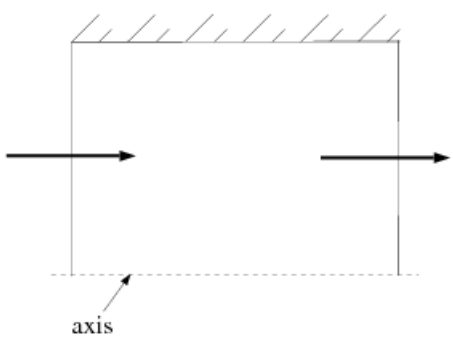
7.3.15.4. Calculation Procedure at Periodic Boundaries

ANSYS FLUENT treats the flow at a periodic boundary as though the opposing periodic plane is a direct neighbor to the cells adjacent to the first periodic boundary. Thus, when calculating the flow through the periodic boundary adjacent to a fluid cell, the flow conditions at the fluid cell adjacent to the opposite periodic plane are used.

7.3.16. Axis Boundary Conditions

The axis boundary type must be used as the centerline of an axisymmetric geometry (see *Figure 7.42* (p. 323)). It can also be used for the centerline of a cylindrical-polar quadrilateral or hexahedral mesh (e.g., a mesh created for a structured-mesh code such as FLUENT 4). You do not need to define any boundary conditions at axis boundaries.

Figure 7.42 Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry



7.3.16.1. Calculation Procedure at Axis Boundaries

To determine the appropriate physical value for a particular variable at a point on the axis, ANSYS FLUENT uses the cell value in the adjacent cell.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.17. Fan Boundary Conditions

The fan model is a lumped parameter model that can be used to determine the impact of a fan with known characteristics upon some larger flow field. The fan boundary type allows you to input an empirical fan curve which governs the relationship between head (pressure rise) and flow rate (velocity) across a fan element. You can also specify radial and tangential components of the fan swirl velocity. The fan model does not provide an accurate description of the detailed flow through the fan blades. Instead, it predicts the amount of flow through the fan. Fans may be used in conjunction with other flow sources, or as the sole source of flow in a simulation. In the latter case, the system flow rate is determined by the balance between losses in the system and the fan curve.

ANSYS FLUENT also provides a connection for a special user-defined fan model that updates the pressure jump function during the calculation. This feature is described in *User-Defined Fan Model* (p. 359).

7.3.17.1. Fan Equations

7.3.17.1.1. Modeling the Pressure Rise Across the Fan

A fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. The relationship may be a constant, a polynomial, piecewise-linear, or piecewise-polynomial function, or a user-defined function.

In the case of a polynomial, the relationship is of the form

$$\Delta p = \sum_{n=1}^{N} f_n v^{n-1} \qquad \textbf{(7–98)}$$

where Δp is the pressure jump, f_n are the pressure-jump polynomial coefficients, and v is the magnitude of the local fluid velocity normal to the fan.

Important

The velocity v can be either positive or negative. You must be careful to model the fan so that a pressure rise occurs for forward flow through the fan.

You can, optionally, use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

7.3.17.1.2. Modeling the Fan Swirl Velocity

For three-dimensional problems, the values of the convected tangential and radial velocity fields can be imposed on the fan surface to generate swirl. These velocities can be specified as functions of the radial distance from the fan center. The relationships may be constant or polynomial functions, or user-defined functions.

Important

You must use SI units for all fan swirl velocity inputs.

For the case of polynomial functions, the tangential and radial velocity components can be specified by the following equations:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$U_\theta = \sum_{n=-1}^{N} f_n r^n ; -1 \le N \le 6 \tag{7–99}$$

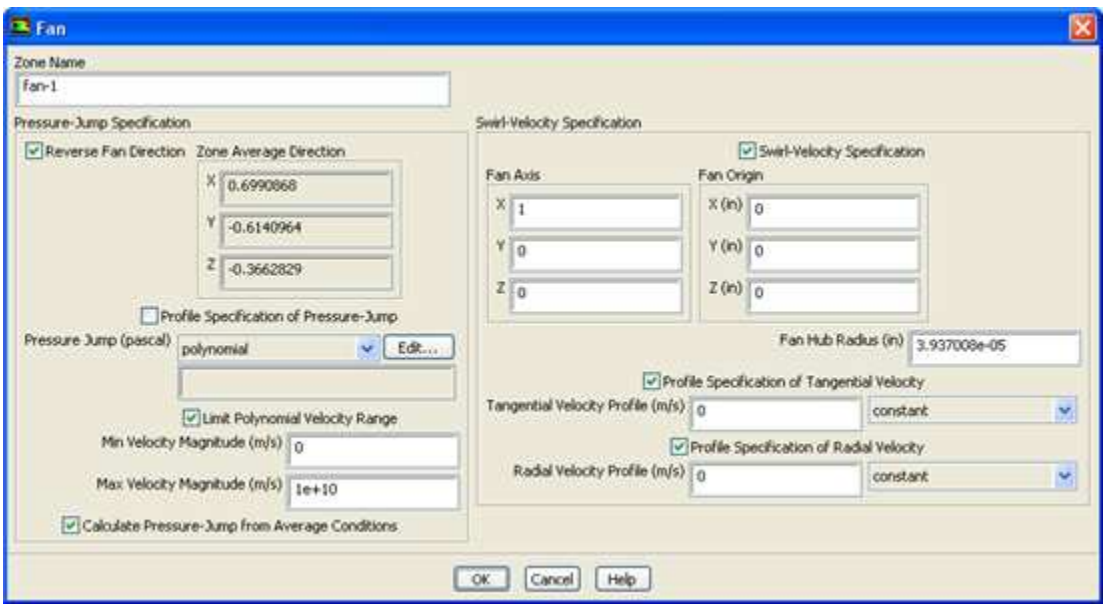
$$U_r = \sum_{n=-1}^{N} g_n r^n ; -1 \le N \le 6 \tag{7–100}$$

where U_θ and U_r are, respectively, the tangential and radial velocities on the fan surface in m/s, f_n and g_n are the tangential and radial velocity polynomial coefficients, and r is the distance to the fan center.

7.3.17.2. User Inputs for Fans

Once the fan zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the fan in the *Fan Dialog Box* (p. 1819) (*Figure 7.43* (p. 325)), which is opened from the *Boundary Conditions Task Page* (p. 1812) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Figure 7.43 The Fan Dialog Box



Inputs for a fan are as follows:

1. Identify the fan zone.
2. Define the pressure jump across the fan.
3. Define the discrete phase boundary condition for the fan (for discrete phase calculations).
4. Define the swirl velocity, if desired (3D only).

7.3.17.2.1. Identifying the Fan Zone

Since the fan is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Thus the fan zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your mesh into ANSYS FLUENT, the fan zone is identified as an **interior** zone, use the **Boundary Conditions** task page (see *Boundary Conditions Task Page* (p. 1812))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(as described in *Changing Cell and Boundary Zone Types* (p. 207)) to change the appropriate **interior** zone to a **fan** zone.

Boundary Conditions

Once the interior zone has been changed to a fan zone, you can open the **Fan** dialog box and specify the pressure jump and, optionally, the swirl velocity.

7.3.17.2.2. Defining the Pressure Jump

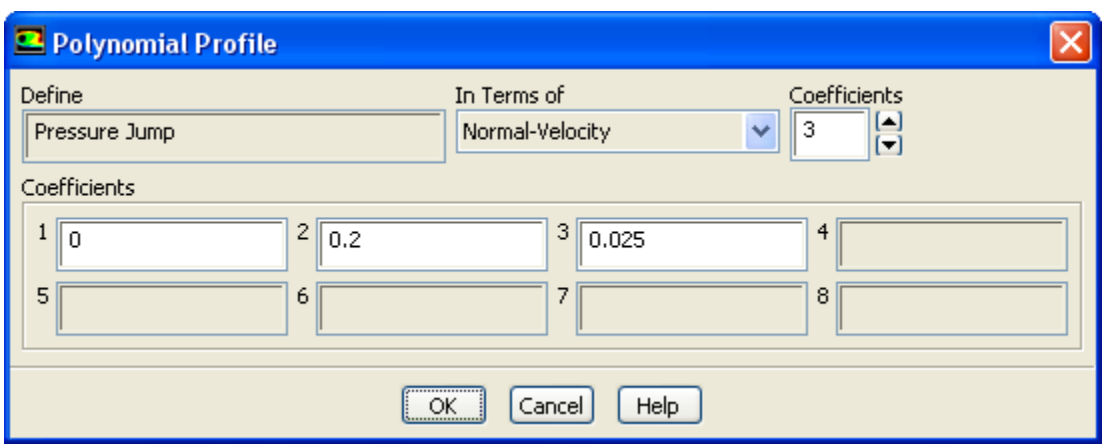
To define the pressure jump, you will specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, a user-defined function, or a constant value. You should also check the **Zone Average Direction** vector to be sure that a pressure rise occurs for forward flow through the fan. The **Zone Average Direction**, calculated by the solver, is the face-averaged direction vector for the fan zone. If this vector is pointing in the direction you want the fan to blow, do not select **Reverse Fan Direction**; if it is pointing in the opposite direction, select **Reverse Fan Direction**.

7.3.17.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure jump:

1. Check that the **Profile Specification of Pressure-Jump** option is off in the *Fan Dialog Box* (p. 1819).
2. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Pressure-Jump**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)
3. In the dialog box that appears for the definition of the **Pressure Jump** function (e.g., *Figure 7.44* (p. 326)), enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) to find out how to use them.

Figure 7.44 Polynomial Profile Dialog Box for Pressure Jump Definition



4. Set any of the optional parameters described below. (optional)

When you define the pressure jump using any of these types of functions, you can choose to limit the minimum and maximum velocity magnitudes used to calculate the pressure jump. Enabling the **Limit Polynomial Velocity Range** option limits the pressure jump when a **Min Velocity Magnitude** and a **Max Velocity Magnitude** are specified.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The values corresponding to the **Min Velocity Magnitude** and the **Max Velocity Magnitude** do not limit the flow field velocity to this range. However, this range does limit the value of the pressure jump, which is a polynomial and a function of velocity, as seen in *Equation 7–98* (p. 324). If the calculated normal velocity magnitude exceeds the **Max Velocity Magnitude** that has been specified, then the pressure jump at the **Max Velocity Magnitude** value will be used. Similarly, if the calculated velocity is less than the specified **Min Velocity Magnitude**, the pressure jump at the **Min Velocity Magnitude** will be substituted for the pressure jump corresponding to the calculated velocity.

You also have the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone. Turning on **Calculate Pressure-Jump from Average Conditions** enables this option.

7.3.17.2.2.2. Constant Value

To define a constant pressure jump, follow these steps:

1. Turn off the **Profile Specification of Pressure-Jump** option in the *Fan Dialog Box* (p. 1819).
2. Choose **constant** in the drop-down list to the right of **Pressure-Jump**.
3. Enter the value for Δp in the **Pressure-Jump** field.

You can follow the procedure below, if it is more convenient:

1. Turn on the **Profile Specification of Pressure-Jump** option.
2. Select **constant** in the drop-down list below **Pressure Jump Profile**, and enter the value for Δp in the **Pressure Jump Profile** field.

7.3.17.2.2.3. User-Defined Function or Profile

For a user-defined pressure-jump function or a function defined in a boundary profile file, you will follow these steps:

1. Turn on the **Profile Specification of Pressure-Jump** option.
2. Choose the appropriate function in the drop-down list below **Pressure Jump Profile**.

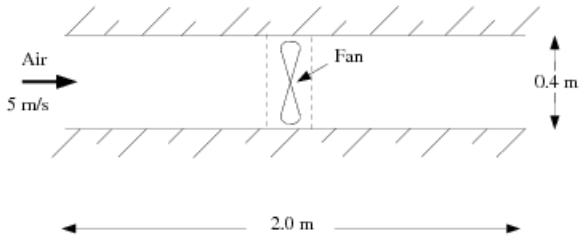
See the UDF Manual for information about user-defined functions, and *Profiles* (p. 366) for details about profile files.

7.3.17.2.2.4. Example: Determining the Pressure Jump Function

This example shows you how to determine the function for the pressure jump. Consider the simple two-dimensional duct flow illustrated in *Figure 7.45* (p. 328). Air at constant density enters the 2.0 m $\times$ 0.4 m duct with a velocity of 15 m/s. Centered in the duct is a fan.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.45 A Fan Located In a 2D Duct



Assume that the fan characteristics are as follows when the fan is operating at 2000 rpm:

Q (m^3/s)	Δp (Pa)
25	0.0
20	175
15	350
10	525
5	700
0	875

where Q is the flow through the fan and Δp is the pressure rise across the fan. The fan characteristics in this example follow a simple linear relationship between pressure rise and flow rate. To convert this into a relationship between pressure rise and velocity, the cross-sectional area of the fan must be known. In this example, assuming that the duct is 1.0 m deep, this area is 0.4 m^2, so that the corresponding velocity values are as follows:

v (m/s)	Δp (Pa)
62.5	0.0
50.0	175
37.5	350
25.0	525
12.5	700
0	875

The polynomial form of this relationship is the following equation for a line:

$$\Delta p = 875 - 14v \tag{7–101}$$

7.3.17.2.3. Defining Discrete Phase Boundary Conditions for the Fan

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the fan. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.17.2.4. Defining the Fan Swirl Velocity

If you want to set tangential and radial velocity fields on the fan surface to generate swirl in a 3D problem, follow these steps:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Turn on the **Swirl-Velocity Specification** option in the *Fan Dialog Box* (p. 1819).
2. Specify the fan's axis of rotation by defining the axis origin (**Fan Origin**) and direction vector (**Fan Axis**).
3. Set the value for the radius of the fan's hub (**Fan Hub Radius**). The default is 1×10^{-6} to avoid division by zero in the polynomial.
4. Set the tangential and radial velocity functions as polynomial functions of radial distance, constant values, or user-defined functions.

Important

You must use SI units for all fan swirl velocity inputs.

7.3.17.2.4.1. Polynomial Function

To define a polynomial function for tangential or radial velocity, follow the steps below:

1. Check that the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option is off in the *Fan Dialog Box* (p. 1819).
2. Enter the coefficients f_n in *Equation 7–99* (p. 325) or g_n in *Equation 7–100* (p. 325) in the **Tangential-** or **Radial-Velocity Polynomial Coefficients** field. Enter f_{-1} first, then f_0, etc. Separate each coefficient by a blank space. Remember that the first coefficient is for $\frac{1}{r}$.

7.3.17.2.4.2. Constant Value

To define a constant tangential or radial velocity, the steps are as follows:

1. Turn on the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option in the *Fan Dialog Box* (p. 1819).
2. Select **constant** in the drop-down list under **Tangential** or **Radial Velocity Profile**.
3. Enter the value for U_θ or U_r in the **Tangential** or **Radial Velocity Profile** field.

You can follow the procedure below, if it is more convenient:

1. Turn off the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option in the *Fan Dialog Box* (p. 1819).
2. Enter the value for U_θ or U_r in the **Tangential-** or **Radial-Velocity Polynomial Coefficients** field.

7.3.17.2.4.3. User-Defined Function or Profile

For a user-defined tangential or radial velocity function or a function contained in a profile file, follow the procedure below:

1. Turn on the **Profile Specification of Tangential** or **Radial Velocity** option.
2. Choose the appropriate function from the drop-down list under **Tangential** or **Radial Velocity Profile**.

See the UDF Manual for information about user-defined functions, and *Profiles* (p. 366) for details about profile files.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.17.3. Postprocessing for Fans

7.3.17.3.1. Reporting the Pressure Rise Through the Fan

You can use the *Surface Integrals Dialog Box* (p. 2031) to report the pressure rise through the fan, as described in *Surface Integration* (p. 1519). There are two steps to this procedure:

1. Create a surface on each side of the fan zone. Use the *Transform Surface Dialog Box* (p. 2139) (as described in *Transforming Surfaces* (p. 1375)) to translate the fan zone slightly upstream and slightly downstream to create two new surfaces.
2. In the **Surface Integrals** dialog box, report the average **Static Pressure** just upstream and just downstream of the fan. You can then calculate the pressure rise through the fan.

7.3.17.3.2. Graphical Plots

Graphical reports of interest with fans are as follows:

- Contours or profiles of **Static Pressure** and **Static Temperature**.
- XY plots of **Static Pressure** and **Static Temperature** vs position.

Displaying Graphics (p. 1379) explains how to generate graphical displays of data.

Important

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the fan. (If you display node values, the cell values on either side of the fan will be averaged to obtain a node value, and you will not see, for example, the pressure jump across the fan.)

7.3.18. Radiator Boundary Conditions

A lumped-parameter model for a heat exchange element (for example, a radiator or condenser), is available in ANSYS FLUENT. The radiator boundary type allows you to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator.

A more detailed heat exchanger model is also available in ANSYS FLUENT. See *Modeling Heat Exchangers* (p. 769) for details.

7.3.18.1. Radiator Equations

7.3.18.1.1. Modeling the Pressure Loss Through a Radiator

A radiator is considered to be infinitely thin, and the pressure drop through the radiator is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient which you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the radiator, v, as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta p = k_L \frac{1}{2} \rho v^2 \quad \textbf{(7–102)}$$

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient, which can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

In the case of a polynomial, the relationship is of the form

$$k_L = \sum_{n=1}^{N} r_n v^{n-1} \quad \textbf{(7–103)}$$

where r_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the radiator.

7.3.18.1.2. Modeling the Heat Transfer Through a Radiator

The heat flux from the radiator to the surrounding fluid is given as

$$q = h\left(T_{air,d} - T_{ext}\right) \quad \textbf{(7–104)}$$

where q is the heat flux, $T_{air,d}$ is the temperature downstream of the heat exchanger (radiator), and T_{ext} is the reference temperature for the liquid. The convective heat transfer coefficient, h, can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

For a polynomial, the relationship is of the form

$$h = \sum_{n=0}^{N} h_n v^n ; 0 \le N \le 7 \quad \textbf{(7–105)}$$

where h_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the radiator in m/s.

Either the actual heat flux (q) or the heat transfer coefficient and radiator temperature (h, T_{ext}) may be specified. q (either the entered value or the value calculated using *Equation 7–104* (p. 331)) is integrated over the radiator surface area.

7.3.18.1.2.1. Calculating the Heat Transfer Coefficient

To model the thermal behavior of the radiator, you must supply an expression for the heat transfer coefficient, h, as a function of the fluid velocity through the radiator, v. To obtain this expression, consider the heat balance equation:

$$q = \frac{\dot{m} c_p \Delta T}{A} = h\left(T_{air,d} - T_{ext}\right) \quad \textbf{(7–106)}$$

where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

q = heat flux (W/m^2)

$\dot{m}$ = fluid mass flow rate (kg/s)

c_p = specific heat capacity of fluid (J/kg-K)

h = empirical heat transfer coefficient (W/m^2-K)

T_{ext} = external temperature (reference temperature for the liquid) (K)

$T_{air,d}$ = temperature downstream from the heat exchanger (K)

A = heat exchanger frontal area (m^2)

Equation 7–106 (p. 331) can be rewritten as

$$q = \frac{\dot{m} c_p \left(T_{air,u} - T_{air,d} \right)}{A} = h \left(T_{air,d} - T_{ext} \right) \tag{7–107}$$

where $T_{air,u}$ is the upstream air temperature. The heat transfer coefficient, h, can therefore be computed as

$$h = \frac{\dot{m} c_p \left(T_{air,u} - T_{air,d} \right)}{A \left(T_{air,d} - T_{ext} \right)} \tag{7–108}$$

or, in terms of the fluid velocity,

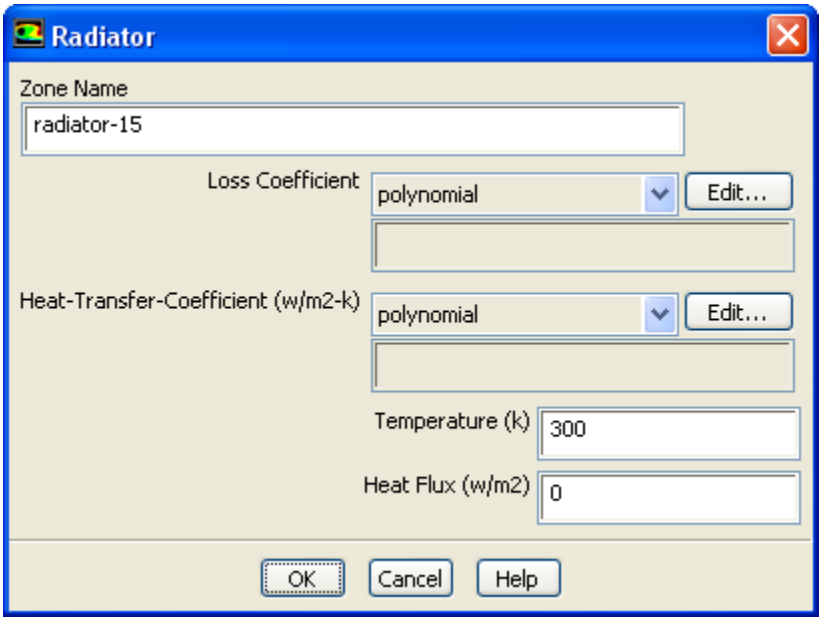
$$h = \frac{\rho v c_p \left(T_{air,u} - T_{air,d} \right)}{T_{air,d} - T_{ext}} \tag{7–109}$$

7.3.18.2. User Inputs for Radiators

Once the radiator zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the radiator in the *Radiator Dialog Box* (p. 1857) (*Figure 7.46* (p. 333)), which is opened from the *Boundary Conditions Task Page* (p. 1812) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.46 The Radiator Dialog Box



The inputs for a radiator are as follows:

1. Identify the radiator zone.
2. Define the pressure loss coefficient.
3. Define either the heat flux or the heat transfer coefficient and radiator temperature.
4. Define the discrete phase boundary condition for the radiator (for discrete phase calculations).

7.3.18.2.1. Identifying the Radiator Zone

Since the radiator is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Thus the radiator zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your mesh into ANSYS FLUENT, the radiator zone is identified as an **interior** zone, use the *Boundary Conditions Task Page* (p. 1812) (as described in *Changing Cell and Boundary Zone Types* (p. 207)) to change the appropriate **interior** zone to a **radiator** zone.

Boundary Conditions

Once the interior zone has been changed to a radiator zone, you can open the **Radiator** dialog box and specify the loss coefficient and heat flux information.

7.3.18.2.2. Defining the Pressure Loss Coefficient Function

To define the pressure loss coefficient k_L you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

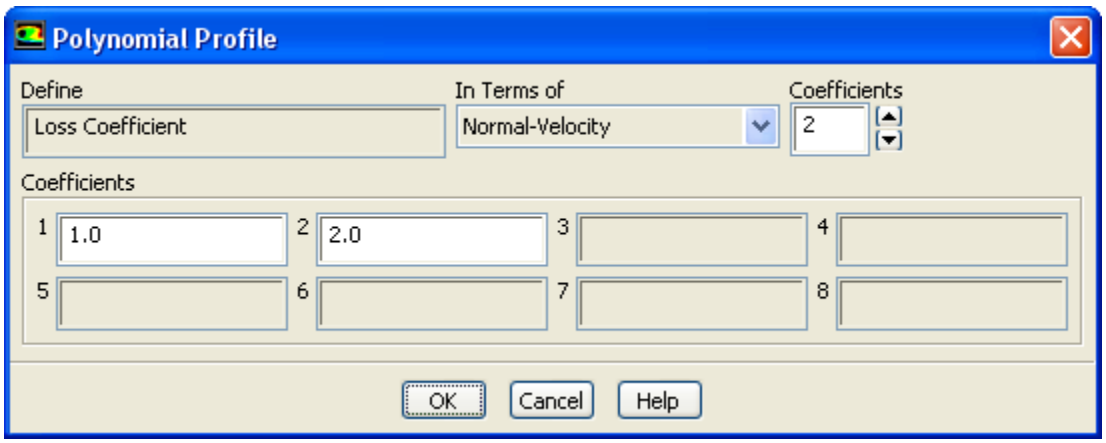
7.3.18.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure loss coefficient:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Loss Coefficient**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)
2. In the dialog box that appears for the definition of the **Loss Coefficient** function (e.g., *Figure 7.47* (p. 334)), enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) to find out how to use them.

Figure 7.47 Polynomial Profile Dialog Box for Loss Coefficient Definition



7.3.18.2.2.2. Constant Value

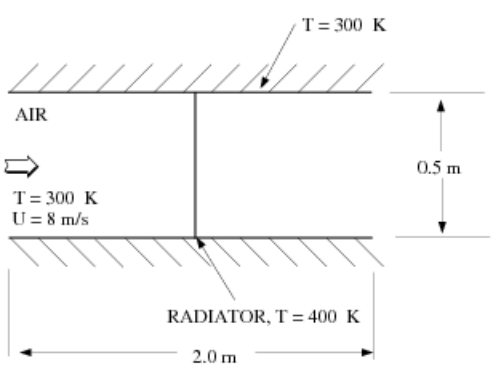
To define a constant loss coefficient, follow these steps:

1. Choose **constant** in the **Loss Coefficient** drop-down list.
2. Enter the value for k_L in the **Loss Coefficient** field.

7.3.18.2.2.3. Example: Calculating the Loss Coefficient

This example shows you how to determine the loss coefficient function. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in *Figure 7.48* (p. 334).

Figure 7.48 A Simple Duct with a Radiator



The radiator characteristics must be known empirically. For this case, assume that the radiator to be modeled yields the test data shown in *Table 7.2: Airside Radiator Data* (p. 335), which was taken with a waterside flow rate of 7 kg/min and an inlet water temperature of 400.0 K. To compute the loss coefficient, it is helpful to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

construct a table with values of the dynamic head, $\frac{1}{2}\rho v^2$, as a function of pressure drop, Δp, and the ratio of these two values, k_L (from *Equation 7–102* (p. 331)). (The air density, defined in *Figure 7.48* (p. 334), is 1.0 kg/m^3.) The reduced data are shown in *Table 7.3: Reduced Radiator Data* (p. 335).

Table 7.2 Airside Radiator Data

Velocity (m/s)	Upstream Temp (K)	Downstream Temp (K)	Pressure Drop
5.0	300.0	330.0	75.0
10.0	300.0	322.5	250.0
15.0	300.0	320.0	450.0

Table 7.3 Reduced Radiator Data

v (m/s)	$\frac{1}{2}\rho v^2$ (Pa)	Δp (Pa)	k_L
5.0	12.5	75.0	6.0
10.0	50.0	250.0	5.0
15.0	112.5	450.0	4.0

The loss coefficient is a linear function of the velocity, decreasing as the velocity increases. The form of this relationship is

$$k_L = 7.0 - 0.2v \qquad \textbf{(7–110)}$$

where v is now the absolute value of the velocity through the radiator.

7.3.18.2.3. Defining the Heat Flux Parameters

As mentioned in *Radiator Equations* (p. 330), you can either define the actual heat flux (q) in the **Heat Flux** field, or set the heat transfer coefficient and radiator temperature (h, T_{ext}). All inputs are in the *Radiator Dialog Box* (p. 1857).

To define the actual heat flux, specify a **Temperature** of `0`, and set the constant **Heat Flux** value.

To define the radiator temperature, enter the value for T_{ext} in the **Temperature** field. To define the heat transfer coefficient, you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

7.3.18.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the heat transfer coefficient:

1. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Heat-Transfer-Coefficient**. (If the function type you want is already selected, you can click on the **Edit...** button to open the dialog box where you will define the function.)
2. In the dialog box that appears for the definition of the **Heat-Transfer-Coefficient** function, enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boxes for temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) to find out how to use them.

7.3.18.2.3.2. Constant Value

To define a constant heat transfer coefficient, follow these steps:

1. Choose **constant** in the **Heat-Transfer-Coefficient** drop-down list.
2. Enter the value for h in the **Heat-Transfer-Coefficient** field.

7.3.18.2.3.3. Example: Determining the Heat Transfer Coefficient Function

This example shows you how to determine the function for the heat transfer coefficient. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in *Figure 7.48* (p. 334).

The data supplied in *Table 7.2: Airside Radiator Data* (p. 335) along with values for the air density (1.0 kg/m^3) and specific heat (1000 J/kg-K) can be used to obtain the following values for the heat transfer coefficient h:

Velocity (m/s)	h **(W/m^2-K)**
5.0	2142.9
10.0	2903.2
15.0	3750.0

The heat transfer coefficient obeys a second-order polynomial relationship (fit to the points in the table above) with the velocity, which is of the form

$$h = 1469.1 + 126.11v + 1.73v^2 \qquad \textbf{(7–111)}$$

Note that the velocity v is assumed to be the absolute value of the velocity passing through the radiator.

7.3.18.2.4. Defining Discrete Phase Boundary Conditions for the Radiator

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the radiator. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.18.3. Postprocessing for Radiators

7.3.18.3.1. Reporting the Radiator Pressure Drop

You can use the *Surface Integrals Dialog Box* (p. 2031) to report the pressure drop across the radiator, as described in *Surface Integration* (p. 1519). There are two steps to this procedure:

1. Create a surface on each side of the radiator zone. Use the *Transform Surface Dialog Box* (p. 2139) (as described in *Transforming Surfaces* (p. 1375)) to translate the radiator zone slightly upstream and slightly downstream to create two new surfaces.
2. In the **Surface Integrals** dialog box, report the average **Static Pressure** just upstream and just downstream of the radiator. You can then calculate the pressure drop across the radiator.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To check this value against the expected value based on *Equation 7–102* (p. 331), you can use the **Surface Integrals** dialog box to report the average normal velocity through the radiator. (If the radiator is not aligned with the x, y, or z axis, you will need to use the *Custom Field Function Calculator Dialog Box* (p. 2106) to generate a function for the velocity normal to the radiator.) Once you have the average normal velocity, you can use *Equation 7–103* (p. 331) to determine the loss coefficient and then *Equation 7–102* (p. 331) to calculate the expected pressure loss.

7.3.18.3.2. Reporting Heat Transfer in the Radiator

To determine the temperature rise across the radiator, follow the procedure outlined above for the pressure drop to generate surfaces upstream and downstream of the radiator. Then use the *Surface Integrals Dialog Box* (p. 2031) (as for the pressure drop report) to report the average **Static Temperature** on each surface. You can then calculate the temperature rise across the radiator.

7.3.18.3.3. Graphical Plots

Graphical reports of interest with radiators are as follows:

- Contours or profiles of **Static Pressure** and **Static Temperature**.
- XY plots of **Static Pressure** and **Static Temperature** vs position.

Displaying Graphics (p. 1379) explains how to generate graphical displays of data.

Important

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the radiator. (If you display node values, the cell values on either side of the radiator will be averaged to obtain a node value, and you will not see, for example, the pressure loss across the radiator.)

7.3.19. Porous Jump Boundary Conditions

Porous jump conditions are used to model a thin "membrane" that has known velocity (pressure-drop) characteristics. It is essentially a 1D simplification of the porous media model available for cell zones. Examples of uses for the porous jump condition include modeling pressure drops through screens and filters, and modeling radiators when you are not concerned with heat transfer. This simpler model should be used whenever possible (instead of the full porous media model) because it is more robust and yields better convergence.

The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy's Law and an additional inertial loss term:

$$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m \qquad \textbf{(7–112)}$$

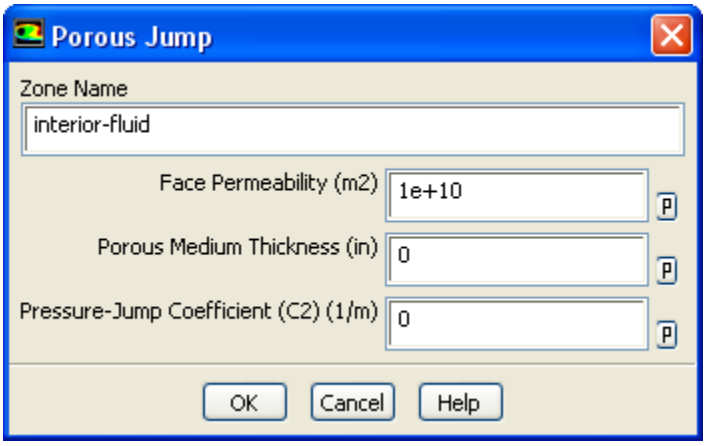
where μ is the laminar fluid viscosity, α is the permeability of the medium, C_2 is the pressure-jump coefficient, v is the velocity normal to the porous face, and Δm is the thickness of the medium. Appropriate values for α and C_2 can be calculated using the techniques described in *User Inputs for Porous Media* (p. 225).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.19.1. User Inputs for the Porous Jump Model

Once the porous jump zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the porous jump in the *Porous Jump Dialog Box* (p. 1842) (*Figure 7.49* (p. 338)), which is opened from the *Boundary Conditions Task Page* (p. 1812) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)).

Figure 7.49 The Porous Jump Dialog Box



The inputs required for the porous jump model are as follows:

1. Identify the porous-jump zone.
2. Set the **Face Permeability** of the medium (α in *Equation 7–112* (p. 337)).
3. Set the **Porous Medium Thickness** (Δm).
4. Set the **Pressure-Jump Coefficient** (C_2).

Note

The unit for the pressure-jump coefficient (C2) is the inverse of length. Should you want to define different units, you can do so by opening the **Units** dialog box and selecting **length-inverse** from the **Quantities** list.

---

5. Define the discrete phase boundary condition for the porous jump (for discrete phase calculations).
6. If you have enabled the solar load model, define the **Solar Boundary Conditions** for the porous jump. These settings allow you to define the boundary such that it acts as a semi-transparent surface (with respect to the solar radiation calculation only), and thus allows a portion of the solar radiation to pass through it. See *Solar Ray Tracing* (p. 754) for details.

7.3.19.1.1. Identifying the Porous Jump Zone

Since the porous jump model is a 1D simplification of the porous media model, the porous-jump zone must be modeled as the interface between cells, rather than a cell zone. Thus the porous-jump zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If the porous-jump zone is not identified as such by default when you read in the mesh (i.e., if it is identified as another type of internal face zone), you can use the *Boundary Conditions Task Page* (p. 1812) to change the appropriate face zone to a **porous-jump** zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define → Boundary Conditions...

The procedure for changing a zone's type is described in *Changing Cell and Boundary Zone Types* (p. 207). Once the zone has been changed to a porous jump, you can open the *Porous Jump Dialog Box* (p. 1842) (as described in *Setting Cell Zone and Boundary Conditions* (p. 208)) and specify the porous jump parameters listed above.

7.3.19.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the porous jump. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

7.3.19.2. Postprocessing for the Porous Jump

Postprocessing suggestions for a problem that includes a porous jump are the same as for porous media problems. See *Postprocessing for Porous Media* (p. 240).

7.4. Non-Reflecting Boundary Conditions

The standard pressure boundary condition, imposed on the boundaries of artificially truncated domain, results in the reflection of the outgoing waves. As a consequence, the interior domain will contain spurious wave reflections. Many applications require precise control of the wave reflections from the domain boundaries to obtain accurate flow solutions. Non-reflecting boundary conditions provide a special treatment to the domain boundaries to control these spurious wave reflections.

In ANSYS FLUENT, two types of non-reflecting boundary conditions (NRBC) are available:

- turbo-specific NRBC
- general NRBC

Turbo-specific NRBC, can be used in both steady and unsteady calculations, and have no geometric restrictions. Both methods are available in the density-based solver and when the compressible ideal-gas law is used.

Important

NRBCs are not available in the pressure-based solver.

Information about non-reflecting boundary conditions (NRBCs) is provided in the following sections.

7.4.1. Turbo-Specific Non-Reflecting Boundary Conditions
7.4.2. General Non-Reflecting Boundary Conditions

7.4.1. Turbo-Specific Non-Reflecting Boundary Conditions

7.4.1.1. Overview

The standard pressure boundary conditions for compressible flow fix specific flow variables at the boundary (e.g., static pressure at an outlet boundary). As a result, pressure waves incident on the boundary will reflect in an unphysical manner, leading to local errors. The effects are more pronounced for internal flow problems where boundaries are usually close to geometry inside the domain, such as compressor or turbine blade rows.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The turbo-specific non-reflecting boundary conditions permit waves to "pass" through the boundaries without spurious reflections. The method used in ANSYS FLUENT is based on the Fourier transformation of solution variables at the non-reflecting boundary [27] (p. 2210). Similar implementations have been investigated by other authors [56] (p. 2212) [72] (p. 2212). The solution is rearranged as a sum of terms corresponding to different frequencies, and their contributions are calculated independently. While the method was originally designed for axial turbomachinery, it has been extended for use with radial turbomachinery.

7.4.1.2. Limitations

Note the following limitations of turbo-specific NRBCs:

- They are available only with the density-based solver (explicit or implicit).
- The current implementation applies to steady compressible flows, with the density calculated using the ideal gas law.
- Inlet and outlet boundary conditions must be pressure inlets and outlets only.

  Important

  Note that the pressure inlet boundaries must be set to the cylindrical coordinate flow specification method when turbo-specific NRBCs are used.

- Quad-mapped (structured) surface meshes must be used for inflow and outflow boundaries in a 3D geometry (i.e., triangular or quad-paved surface meshes are not allowed). See *Figure 7.50* (p. 341) and *Figure 7.51* (p. 341) for examples.

  Important

  Note that you may use unstructured meshes in 2D geometries (*Figure 7.52* (p. 342)), and an unstructured mesh may be used away from the inlet and outlet boundaries in 3D geometries.

- The turbo-specific NRBC [27] (p. 2210) has been extended for use on 3D geometries [72] (p. 2212) by decoupling the tangential flow variations from the radial variations. This approximation works best for geometries with a blade pitch that is small compared to the radius of the geometry.
- Reverse flow on the inflow and outflow boundaries are not allowed. If strong reverse flow is present, then you should consider using the General NRBCs instead.
- NRBCs are not compatible with species transport models. They are mainly used to solve ideal-gas single-species flow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.50 Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem

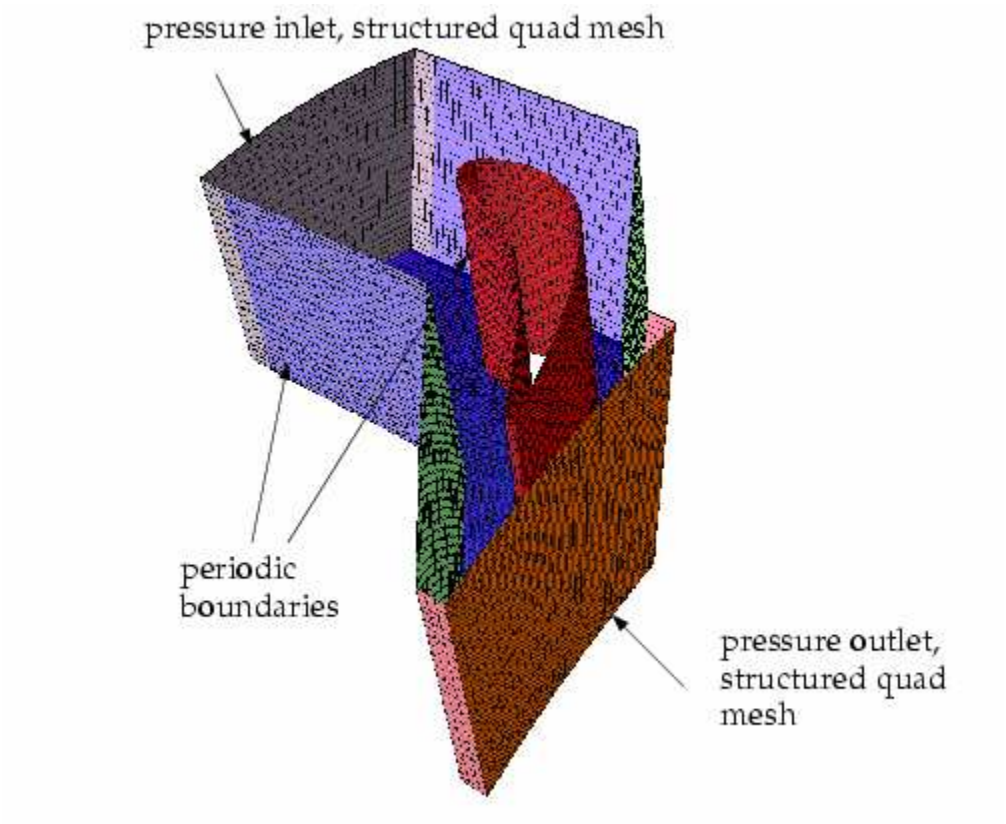
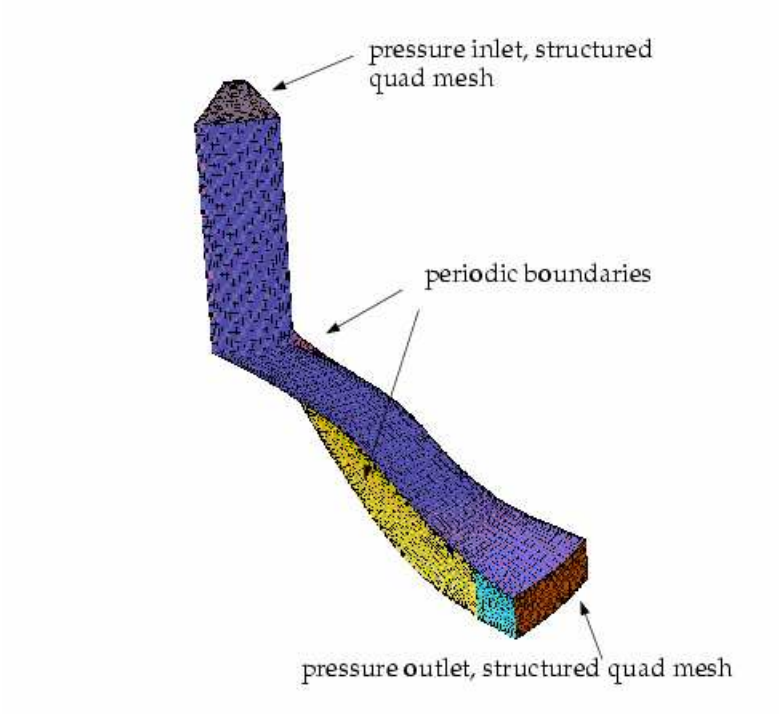
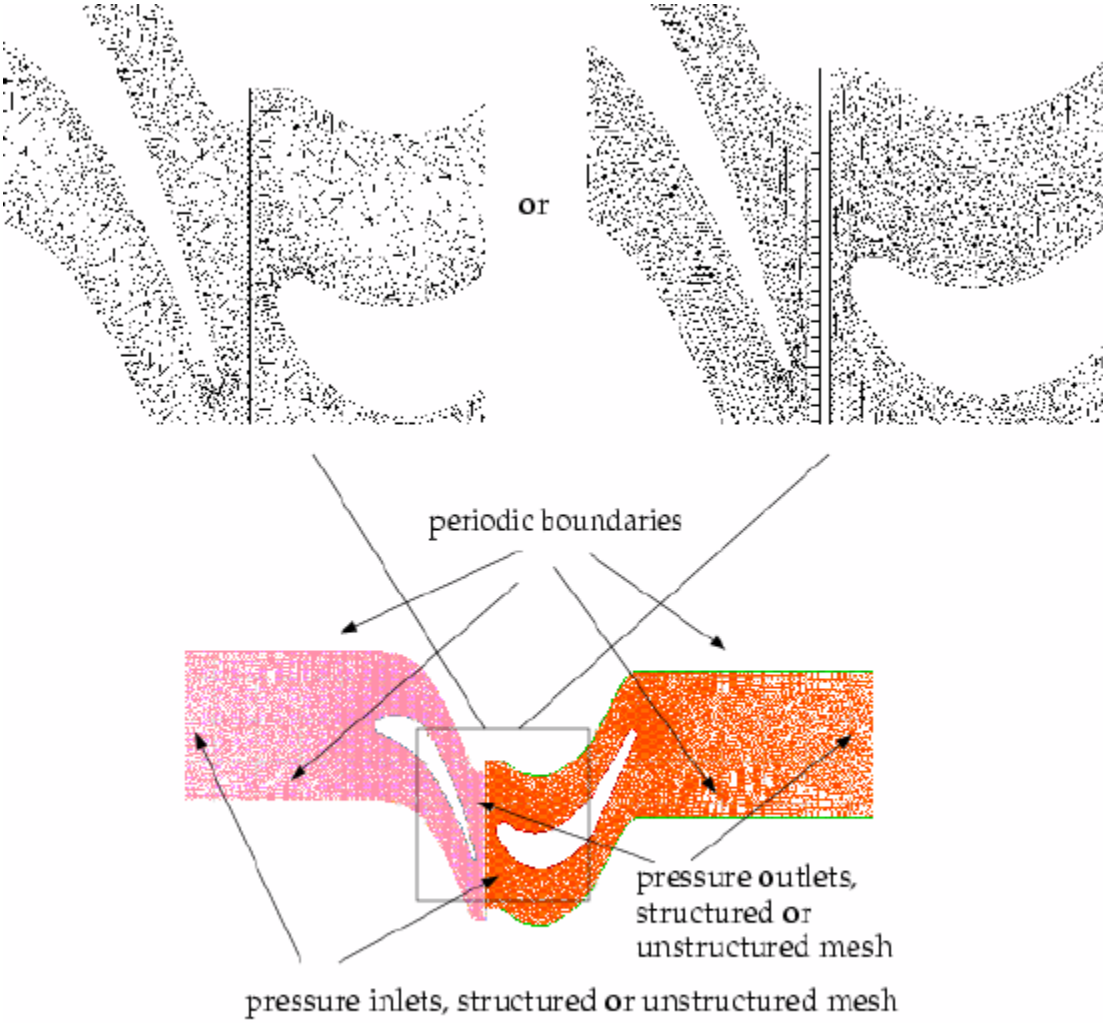


Figure 7.51 Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.52 Mesh and Prescribed Boundary Conditions in a 2D Case



7.4.1.3. Theory

Turbo-specific NRBCs are based on Fourier decomposition of solutions to the linearized Euler equations. The solution at the inlet and outlet boundaries is circumferentially decomposed into Fourier modes, with the 0th mode representing the average boundary value (which is to be imposed as a user input), and higher harmonics that are modified to eliminate reflections [*72*] (p. 2212).

7.4.1.3.1. Equations in Characteristic Variable Form

In order to treat individual waves, the linearized Euler equations are transformed to characteristic variable (C_i) form. If we first consider the 1D form of the linearized Euler equations, it can be shown that the characteristic variables C_i are related to the solution variables as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tilde{Q}=T^{-1}C \tag{7–113}$$

where

$$\tilde{Q}=\begin{Bmatrix} \tilde{\rho} \\ \tilde{u}_a \\ \tilde{u}_t \\ \tilde{u}_r \\ \tilde{p} \end{Bmatrix}, T^{-1}=\begin{bmatrix} -\frac{1}{\overline{a}^2} & 0 & 0 & \frac{1}{2\overline{a}^2} & \frac{1}{2\overline{a}^2} \\ 0 & 0 & 0 & \frac{1}{2\,\overline{\rho}\,\overline{a}} & \frac{1}{2\,\overline{\rho}\,\overline{a}} \\ 0 & \frac{1}{\overline{\rho}\,\overline{a}} & 0 & 0 & 0 \\ 0 & 0 & \frac{1}{\overline{\rho}\,\overline{a}} & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2} & \frac{1}{2} \end{bmatrix}, C=\begin{Bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \\ C_5 \end{Bmatrix} \tag{7–114}$$

where $\overline{a}$ is the average acoustic speed along a boundary zone, $\tilde{\rho}$, $\tilde{u}_a$, $\tilde{u}_t$, $\tilde{u}_r$, and $\tilde{p}$ represent perturbations from a uniform condition (e.g., $\tilde{\rho}=\rho-\overline{\rho}$,$\tilde{p}=p-\overline{p}$, etc.).

Note that the analysis is performed using the cylindrical coordinate system. All overlined (averaged) flow field variables (e.g., $\overline{\rho}$, $\overline{a}$) are intended to be averaged along the pitchwise direction.

In quasi-3D approaches [*27*] (p. 2210) [*56*] (p. 2212) [*72*] (p. 2212), a procedure is developed to determine the changes in the characteristic variables, denoted by δC_i, at the boundaries such that waves will not reflect. These changes in characteristic variables are determined as follows:

$$\delta C=T\ \delta Q \tag{7–115}$$

where

$$\delta C=\begin{Bmatrix} \delta C_1 \\ \delta C_2 \\ \delta C_3 \\ \delta C_4 \\ \delta C_5 \end{Bmatrix}, T=\begin{bmatrix} -\overline{a}^2 & 0 & 0 & 0 & 1 \\ 0 & 0 & \overline{\rho}\ \overline{a} & 0 & 0 \\ 0 & 0 & 0 & \overline{\rho}\ \overline{a} & 0 \\ 0 & \overline{\rho}\ \overline{a} & 0 & 0 & 1 \\ 0 & -\overline{\rho}\ \overline{a} & 0 & 0 & 1 \end{bmatrix}, \delta Q=\begin{Bmatrix} \delta\rho \\ \delta u_a \\ \delta u_t \\ \delta u_r \\ \delta p \end{Bmatrix} \tag{7–116}$$

The changes to the outgoing characteristics — one characteristic for subsonic inflow (δC_5), and four characteristics for subsonic outflow ($\delta C_1, \delta C_2, \delta C_3, \delta C_4$) — are determined from extrapolation of the flow field variables using *Equation 7–115* (p. 343).

The changes in the incoming characteristics — four characteristics for subsonic inflow ($\delta C_1, \delta C_2, \delta C_3, \delta C_4$), and one characteristic for subsonic outflow (δC_5) — are split into two components: average change along

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the boundary ($\delta\overline{C}_i$), and local changes in the characteristic variable due to harmonic variation along the boundary (δC_{iL}). The incoming characteristics are therefore given by

$$\delta C_{i_j} = \delta C_{iold_j} + \sigma\left(\delta C_{inew_j} - \delta C_{iold_j}\right) \tag{7–117}$$

$$\delta C_{inew_j} = \left(\delta\overline{C}_i + \delta C_{iL_j}\right) \tag{7–118}$$

where $i = 1, 2, 3, 4$ on the inlet boundary or $i = 5$ on the outlet boundary, and $j = 1, \ldots, N$ is the grid index in the pitchwise direction including the periodic point once. The under-relaxation factor σ has a default value of 0.75. Note that this method assumes a periodic solution in the pitchwise direction.

The flow is decomposed into mean and circumferential components using Fourier decomposition. The 0th Fourier mode corresponds to the average circumferential solution, and is treated according to the standard 1D characteristic theory. The remaining parts of the solution are described by a sum of harmonics, and treated as 2D non-reflecting boundary conditions [*27*] (p. 2210).

7.4.1.3.2. Inlet Boundary

For subsonic inflow, there is one outgoing characteristic (δC_5) determined from *Equation 7–115* (p. 343), and four incoming characteristics ($\delta C_1, \delta C_2, \delta C_3, \delta C_4$) calculated using *Equation 7–117* (p. 344). The average changes in the incoming characteristics are computed from the requirement that the entropy (s), radial and tangential flow angles (α_r and α_t), and stagnation enthalpy (h_0) are specified. Note that in ANSYS FLUENT you can specify p_0 and T_0 at the inlet, from which s_{in} and $h_{0_{in}}$ are easily obtained. This is equivalent to forcing the following four residuals to be zero:

$$R_1 = \overline{\rho}(\overline{s} - s_{in}) \tag{7–119}$$

$$R_2 = \overline{\rho}\ \overline{a}(\overline{u}_t - \overline{u}_a \tan\alpha_t) \tag{7–120}$$

$$R_3 = \overline{\rho}\ \overline{a}(\overline{u}_r - \overline{u}_a \tan\alpha_r) \tag{7–121}$$

$$R_4 = \overline{\rho}\left(\overline{h}_0 - h_{0_{in}}\right) \tag{7–122}$$

where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$s_{in} = \gamma \ln\left(T_{0_{in}}\right) - (\gamma - 1) \ln\left(p_{0_{in}}\right) \tag{7–123}$$

$$h_{0_{in}} = c_p T_{0_{in}} \tag{7–124}$$

The average characteristic is then obtained from residual linearization as follows (see also *Figure 7.53* (p. 346)

$$\begin{Bmatrix} \delta \overline{C}_1 \\ \delta \overline{C}_2 \\ \delta \overline{C}_3 \\ \delta \overline{C}_4 \end{Bmatrix} = \begin{bmatrix} -1 & 0 & 0 & 0 \\ \frac{\tan\alpha_t}{(1-\gamma)M} & \frac{M_1}{M} & \frac{-M_t}{M}\tan\alpha_t & \frac{\tan\alpha_t}{M} \\ \frac{\tan\alpha_r}{(\gamma-1)M} & \frac{M_t}{M}\tan\alpha_t & \frac{M_2}{M} & \frac{-\tan\alpha_r}{M} \\ \frac{2}{(\gamma-1)M} & 2\frac{M_t}{M} & 2\frac{M_r}{M} & \frac{-2}{M} \end{bmatrix} \begin{Bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \end{Bmatrix} \tag{7–125}$$

where

$$M_a = \frac{\overline{u}_a}{\overline{a}} \tag{7–126}$$

$$M_t = \frac{\overline{u}_t}{\overline{a}} \tag{7–127}$$

$$M_r = \frac{\overline{u}_r}{\overline{a}} \tag{7–128}$$

and

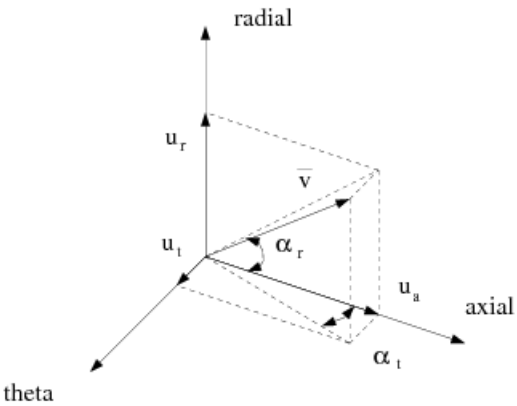
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$M = 1 + M_a - M_t \tan\alpha_t + M_r \tan\alpha_r \quad \textbf{(7–129)}$$

$$M_1 = -1 - M_a - M_r \tan\alpha_r \quad \textbf{(7–130)}$$

$$M_2 = -1 - M_a - M_t \tan\alpha_t \quad \textbf{(7–131)}$$

Figure 7.53 Prescribed Inlet Angles



where

$$|v| = \sqrt{u_t^2 + u_r^2 + u_a^2} \quad \textbf{(7–132)}$$

$$e_t = \frac{u_t}{|v|} \quad \textbf{(7–133)}$$

$$e_r = \frac{u_r}{|v|} \quad \textbf{(7–134)}$$

$$e_a = \frac{u_a}{|v|} \quad \textbf{(7–135)}$$

$$\tan\alpha_t = \frac{e_t}{e_a} \quad \textbf{(7–136)}$$

$$\tan\alpha_r = \frac{e_r}{e_a} \quad \textbf{(7–137)}$$

To address the local characteristic changes at each j grid point along the inflow boundary, the following relations are developed [*27*] (p. 2210) [*72*] (p. 2212):

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{aligned}
\delta C_{1L_j} &= \overline{p}\left(s_j - \overline{s}\right) \\
\delta C_{2L_j} &= C'_{2_j} - \overline{\rho}\,\overline{a}\left(u_{t_j} - \overline{u}_t\right) \\
\delta C_{3L_j} &= -\overline{\rho}\,\overline{a}\left(u_{r_j} - \overline{u}_r\right) \\
\delta C_{4L_j} &= \frac{-2}{\left(1+M_{a_j}\right)}\left(\frac{1}{\gamma-1}\delta C_{1L_j} + M_{t_j}\delta C_{2L_j} + M_{r_j}\delta C_{3L_j} + \overline{\rho}\left(h_{0_j} - \overline{h}_0\right)\right)
\end{aligned} \tag{7–138}$$

Note that the relation for the first and fourth local characteristics force the local entropy and stagnation enthalpy to match their average steady-state values.

The characteristic variable C'_{2_j} is computed from the inverse discrete Fourier transform of the second characteristic. The discrete Fourier transform of the second characteristic in turn is related to the discrete Fourier transform of the fifth characteristic. Hence, the characteristic variable C'_{2_j} is computed along the pitch as follows:

$$C'_{2_j} = 2\Re\left(\sum_{n=1}^{\frac{N}{2}-1} \hat{C}_{2_n} \exp\left(i2\pi n \frac{\theta_j - \theta_1}{\theta_N - \theta_1}\right)\right) \tag{7–139}$$

The Fourier coefficients C'_{2_n} are related to a set of equidistant distributed characteristic variables $C^*_{5_j}$ by the following [56] (p. 2212):

$$\hat{C}_{2_n} = \begin{cases} -\dfrac{\overline{u}_t + B}{N(\overline{a} + \overline{u}_a)} \displaystyle\sum_{j=1}^{N} C^*_{5_j} \exp\left(-i2\pi \frac{jn}{N}\right) & \beta > 0 \\ -\dfrac{\overline{u}_t + B}{\overline{a} + \overline{u}_a} C_{5_j} & \beta < 0 \end{cases} \tag{7–140}$$

where

$$B = \begin{cases} i\sqrt{\beta} & \beta > 0 \\ -sign(\overline{u}_t)\sqrt{|\beta|} & \beta < 0 \end{cases} \tag{7–141}$$

and

$$\beta = \overline{a}^2 - \overline{u}_a^2 - \overline{u}_t^2 \tag{7–142}$$

The set of equidistributed characteristic variables $C^*_{5_j}$ is computed from arbitrary distributed C_{5_j} by using a cubic spline for interpolation, where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$C_{5_j} = -\overline{\rho}\ \overline{a}\left(u_{a_j} - \overline{u}_a\right) + \left(p_j - \overline{p}\right) \tag{7–143}$$

For supersonic inflow the user-prescribed static pressure ($p_{s_{in}}$) along with total pressure ($p_{0_{in}}$) and total temperature ($T_{0_{in}}$) are sufficient for determining the flow condition at the inlet.

7.4.1.3.3. Outlet Boundary

For subsonic outflow, there are four outgoing characteristics ($\delta C_1, \delta C_2, \delta C_3$, and δC_4) calculated using *Equation 7–115* (p. 343), and one incoming characteristic (δC_5) determined from *Equation 7–117* (p. 344). The average change in the incoming fifth characteristic is given by

$$\delta \overline{C}_5 = -2\left(\overline{p} - p_{out}\right) \tag{7–144}$$

where $\overline{p}$ is the current averaged pressure at the exit plane and p_{out} is the desirable average exit pressure (this value is specified by you for single-blade calculations or obtained from the assigned profile for mixing-plane calculations). The local changes (δC_{5L_j}) are given by

$$\delta C_{5L_j} = C'_{5_j} + \overline{\rho}\ \overline{a}\left(u_{a_j} - \overline{u}_a\right) - \left(p_j - \overline{p}\right) \tag{7–145}$$

The characteristic variable C'_{5_j} is computed along the pitch as follows:

$$C'_{5_j} = 2\Re\left(\sum_{n=1}^{\frac{N}{2}-1} \hat{C}_{5_n} \exp\left(i2\pi n \frac{\theta_j - \theta_1}{\theta_N - \theta_1}\right)\right) \tag{7–146}$$

The Fourier coefficients $\hat{C}_{5_n}$ are related to two sets of equidistantly distributed characteristic variables ($C^*_{2_j}$ and $C^*_{4_j}$, respectively) and given by the following [56] (p. 2212):

$$\hat{C}_{5_n} = \begin{cases} \dfrac{A_2}{N}\displaystyle\sum_{j=1}^{N} C^*_{2_j} \exp\left(i2\pi \frac{jn}{N}\right) - \dfrac{A_4}{N}\displaystyle\sum_{j=1}^{N} C^*_{4_j} \exp\left(i2\pi \frac{jn}{N}\right) & \beta > 0 \\ A_2 C_{2_j} - A_4 C_{4_j} & \beta < 0 \end{cases} \tag{7–147}$$

where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$A_2 = \frac{2\bar{u}_a}{B - \bar{u}_t} \tag{7–148}$$

$$A_4 = \frac{B + \bar{u}_t}{B - \bar{u}_t} \tag{7–149}$$

The two sets of equidistributed characteristic variables ($C_{2_j}^*$ and $C_{4_j}^*$) are computed from arbitrarily distributed C_{2_j} and C_{4_j} characteristics by using a cubic spline for interpolation, where

$$C_{2_j} = \bar{\rho}\ \bar{a}\left(u_{t_j} - \bar{u}_t\right) \tag{7–150}$$

$$C_{4_j} = \bar{\rho}\ \bar{a}\left(u_{a_j} - \bar{u}_a\right) + \left(p_j - \bar{p}\right) \tag{7–151}$$

For supersonic outflow all flow field variables are extrapolated from the interior.

7.4.1.3.4. Updated Flow Variables

Once the changes in the characteristics are determined on the inflow or outflow boundaries, the changes in the flow variables δQ can be obtained from *Equation 7–115* (p. 343). Therefore, the values of the flow variables at the boundary faces are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$p_f = p_j + \delta p \tag{7–152}$$

$$u_{a_f} = u_{a_j} + \delta u_a \tag{7–153}$$

$$u_{t_f} = u_{t_j} + \delta u_t \tag{7–154}$$

$$u_{r_f} = u_{r_j} + \delta u_r \tag{7–155}$$

$$T_f = T_j + \delta T \tag{7–156}$$

7.4.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions

Important

If you intend to use turbo-specific NRBCs in conjunction with the density-based implicit solver, it is recommended that you first converge the solution before turning on turbo-specific NRBCs, then converge it again with turbo-specific NRBCs turned on. If the solution is diverging, then you should lower the CFL number. These steps are necessary because only approximate flux Jacobians are used for the pressure-inlet and pressure-outlet boundaries when turbo-specific NRBCs are activated with the density-based implicit solver.

The procedure for using the turbo-specific NRBCs is as follows:

1. Turn on the turbo-specific NRBCs using the `non-reflecting-bc` text command:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `enable?`

   If you are not sure whether or not NRBCs are turned on, use the `show-status` text command.

2. Perform NRBC initialization using the `initialize` text command:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `initialize`

   If the initialization is successful, a summary printout of the domain extent will be displayed. If the initialization is not successful, an error message will be displayed indicating the source of the problem. The initialization will set up the pressure-inlet and pressure-outlet boundaries for use with turbo-specific NRBCs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the pressure inlet boundaries must be set to the cylindrical coordinate flow specification method when turbo-specific NRBCs are used.

3. If necessary, modify the parameters in the `set/` submenu:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `set`

   `under-relaxation`
   allows you to set the value of the under-relaxation factor σ in *Equation 7–117* (p. 344). The default value is 0.75.

   `discretization`
   allows you to set the discretization scheme. The default is to use higher-order reconstruction if available.

   `verbosity`
   allows you to control the amount of information printed to the console during an NRBC calculation.

   - `0` : silent
   - `1` : basic information (default)
   - `2`: detailed information (for debugging purposes only)

7.4.1.4.1. Using the NRBCs with the Mixing-Plane Model

If you want to use the NRBCs with the mixing-plane model you must define the mixing plane interfaces as pressure-outlet and pressure-inlet zone type pairs.

Important

Turbo-specific NRBCs should not be used with the mixing-plane model if reverse flow is present across the mixing-plane.

7.4.1.4.2. Using the NRBCs in Parallel ANSYS FLUENT

When the turbo-specific NRBCs are used in conjunction with the parallel solver, all cells in each boundary zone, where NRBCs will be applied, must be located or contained within a single partition. You can ensure this by manually partitioning the mesh (see *Partitioning the Mesh Manually and Balancing the Load* (p. 1605) for more information).

7.4.2. General Non-Reflecting Boundary Conditions

7.4.2.1. Overview

The general non-reflecting boundary conditions in ANSYS FLUENT are based on characteristic wave relations derived from the Euler equations, and applied only on pressure-outlet boundary conditions. To obtain the primitive flow quantities (P, u, v, w, T) on the pressure-outlet, reformulated Euler equations are solved on the boundary of the domain in an algorithm similar to the flow equations applied to the interior of the domain.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unlike the turbo-specific NRBC method is not restricted by geometric constraints or mesh type. However, good cell skewness near the boundaries where the NRBCs can be applied to steady or transient flows as long as the compressible ideal-gas law is used.

7.4.2.2. Restrictions and Limitations

Note the following restrictions and limitations on the general NRBCs:

- The general NRBC is available only with the density-based solver.
- The general NRBC is available only with compressible flow while using the ideal-gas law.

Important

The general NRBC should not be used with the wet steam or real gas models.

- The general NRBC is not available if the target mass flow rate is activated in the pressure-outlet dialog box.
- The general NRBC is activated.

Important

If you switch from Turbo-specific NRBC or vice versa, then make sure you switch off one NRBC models at the same time.

- NRBCs are not compatible with species transport models. They are mainly used to solve ideal-gas single-species flow.

7.4.2.3. Theory

General NRBCs are derived by first recasting the Euler equations in an orthogonal coordinate system (x_1, x_2, x_3) such that one of the coordinates, x_1, is normal to the boundary *Figure 7.54* (p. 354). The characteristic analysis [*96*] (p. 2214) [*97*] (p. 2214) is then used to modify terms corresponding to waves propagating in the x_1 normal direction. When doing so, a system of equations can be written to describe the wave propagation as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$
\begin{aligned}
\frac{\partial \rho}{\partial t} &+ d_1 + \frac{\partial m_2}{\partial x_2} + \frac{\partial m_3}{\partial x_3} = 0 \\
\frac{\partial m_1}{\partial t} &+ U_1 d_1 + \rho d_3 + \frac{\partial (m_1 U_2)}{\partial x_2} + \frac{\partial (m_1 U_3)}{\partial x_3} = 0 \\
\frac{\partial m_2}{\partial t} &+ U_2 d_1 + \rho d_4 + \frac{\partial (m_2 U_2)}{\partial x_2} + \frac{\partial (m_2 U_3)}{\partial x_3} + \frac{\partial P}{\partial x_2} = 0 \\
\frac{\partial m_3}{\partial t} &+ U_3 d_1 + \rho d_5 + \frac{\partial (m_3 U_2)}{\partial x_2} + \frac{\partial (m_3 U_3)}{\partial x_3} + \frac{\partial P}{\partial x_3} = 0 \\
\frac{\partial \rho E}{\partial t} &+ \frac{1}{2} |V|^2 d_1 + \frac{d_2}{(\gamma - 1)} + m_1 d_3 + m_2 d_4 + m_3 d_5 + \frac{\partial [(\rho E + P) U_2]}{\partial x_2} + \frac{\partial [(\rho E + P) U_3]}{\partial x_3} = 0
\end{aligned}
\tag{7–157}
$$

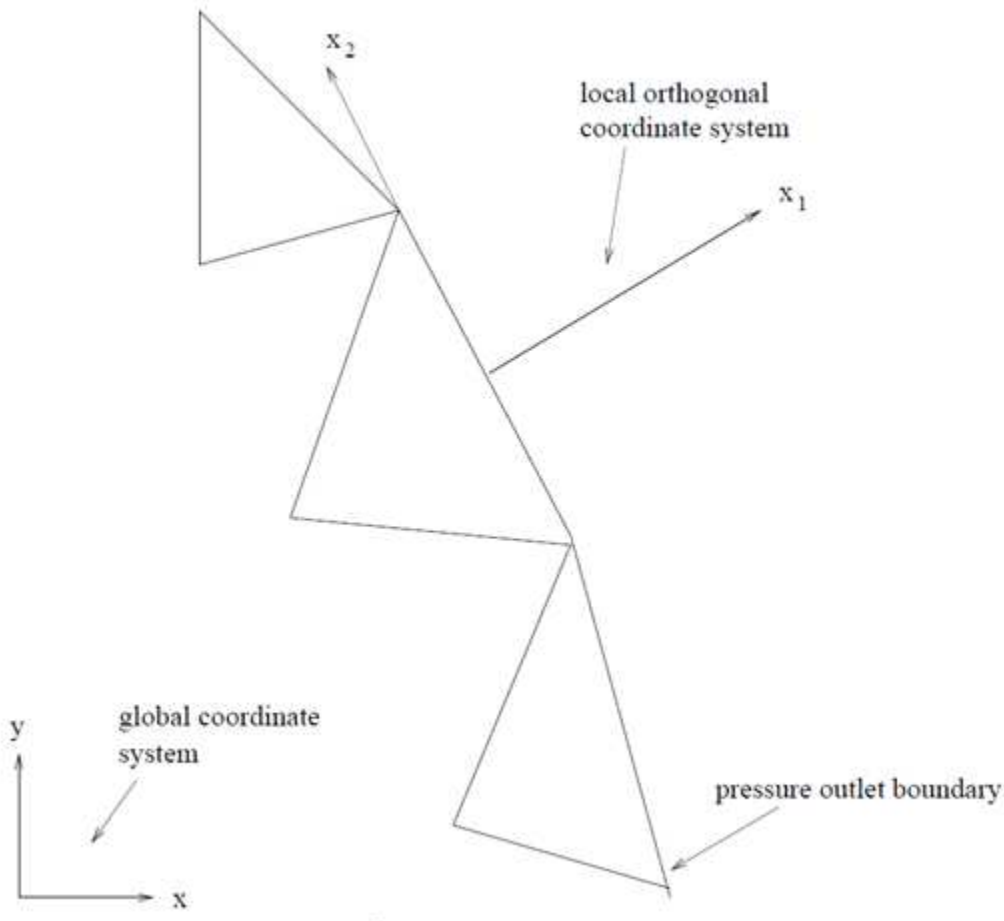
Where $m_1 = \rho U_1$, $m_2 = \rho U_2$ and $m_3 = \rho U_3$ and U_1, U_2 and U_3 are the velocity components in the coordinate system (x_1, x_2, x_3). The equations above are solved on pressure-outlet boundaries, along with the interior governing flow equations, using similar time stepping algorithms to obtain the values of the primitive flow variables (P, u, v, w, T).

Important

Note that a transformation between the local orthogonal coordinate system (x_1, x_2, x_3) and the global Cartesian system (X, Y, Z) must be defined on each face on the boundary to obtain the velocity components (u, v, w) in a global Cartesian system.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.54 The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method



The d_i terms in the transformed Euler equations contain the outgoing and incoming characteristic wave amplitudes, L_i, and are defined as follows:

$$
\begin{aligned}
d_1 &= \frac{1}{c^2}\left[L_2 + \frac{1}{2}(L_5 + L_1) \right] \\
d_2 &= \frac{1}{2}(L_5 + L_1) \\
d_3 &= \frac{1}{2\rho c}(L_5 - L_1) \\
d_4 &= L_3 \\
d_5 &= L_4
\end{aligned}
\tag{7–158}
$$

From characteristic analyses, the wave amplitudes, L_i, are given by:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

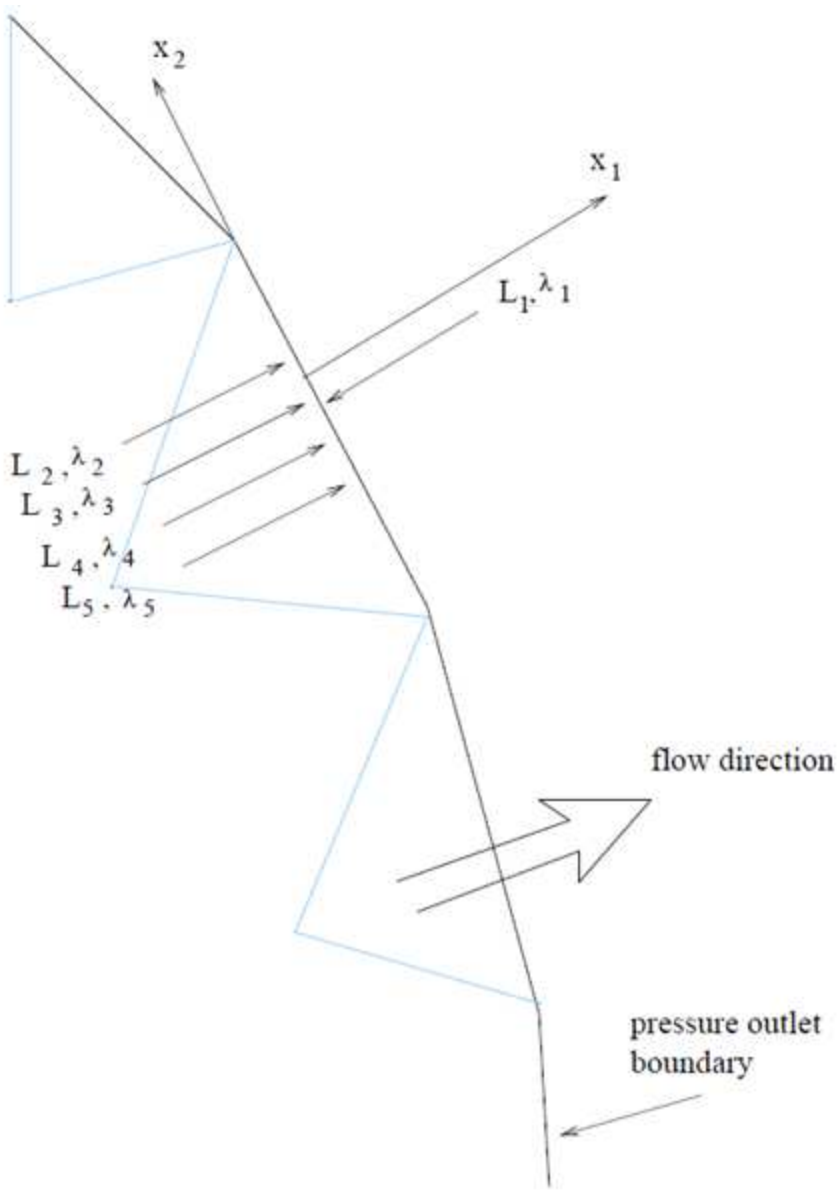
$$
\begin{aligned}
L_1 &= \lambda_1 \left(\frac{\partial P}{\partial x_1} - \rho c \frac{\partial U_1}{\partial x_1} \right) \\
L_2 &= \lambda_2 \left(c^2 \frac{\partial P}{\partial x_1} - \frac{\partial P}{\partial x_1} \right) \\
L_3 &= \lambda_3 \frac{\partial U_2}{\partial x_1} \\
L_4 &= \lambda_4 \frac{\partial U_3}{\partial x_1} \\
L_5 &= \lambda_5 \left(\frac{\partial P}{\partial x_1} + \rho c \frac{\partial U_1}{\partial x_1} \right)
\end{aligned}
\tag{7–159}
$$

The outgoing and incoming characteristic waves are associated with the characteristic velocities of the system (i.e eigenvalues), λ_i, as seen in *Figure 7.55* (p. 356). These eigenvalues are given by:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{aligned}\lambda_1 &= U_1 - c \\ \lambda_2 = \lambda_3 = \lambda_4 &= U_1 \\ \lambda_5 &= U_1 + c\end{aligned} \tag{7–160}$$

Figure 7.55 Waves Leaving and Entering a Boundary Face on a Pressure-Outlet Boundary. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition



For subsonic flow leaving a pressure-outlet boundary, four waves leave the domain (associated with positive eigenvalues λ_2, λ_3, λ_4, and λ_5) and one enters the domain (associated with negative eigenvalue λ_1).

To solve *Equation 7–157* (p. 353) on a pressure-outlet boundary, the values of L_2, L_3, L_4 and L_5 must be first determined from *Equation 7–159* (p. 355) by using extrapolated values of $\frac{\partial P}{\partial x_1}$, $\frac{\partial U_1}{\partial x_1}$, $\frac{\partial U_2}{\partial x_1}$, and $\frac{\partial U_3}{\partial x_1}$ from inside the domain. Then, for the lone incoming wave, the Linear Relaxation Method (LRM) of Poinsot [*65*] (p. 2212)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[66] (p. 2212) is used to determine the value of the L_1 wave amplitude. The LRM method sets the value of the incoming wave amplitude to be proportional to the differences between the local pressure on a boundary face and the imposed exit pressure. Therefore, L_1 is given by

$$L_1 = K\,(P - P_{exit}) \tag{7–161}$$

where P_{exit} is the imposed pressure at the exit boundary, K is the relaxation factor, and P is the local pressure value at the boundary.

In general, the desirable average pressure on a non-reflecting boundary can be either relaxed toward a pressure value at infinity or enforced to be equivalent to some desired pressure at the exit of the boundary.

If you want the average pressure at the boundary to relax toward P at infinity (i.e. $P_{exit} = P_\infty$), the suggested K factor is given by:

$$K = \sigma_1 \left(1 - M_{max}{}^2\right) \frac{c}{h} \tag{7–162}$$

where c is the acoustic speed, h is the domain size, M_{max} is the maximum Mach number in the domain, and σ_1 is the under-relaxation factor (default value is 0.15). On the other hand, if the desired average pressure at the boundary is to approach a specific imposed value at the boundary, then the K factor is given by:

$$K = \sigma_2 c \tag{7–163}$$

where the default value for σ_2 is 5.0

7.4.2.4. Using General Non-Reflecting Boundary Conditions

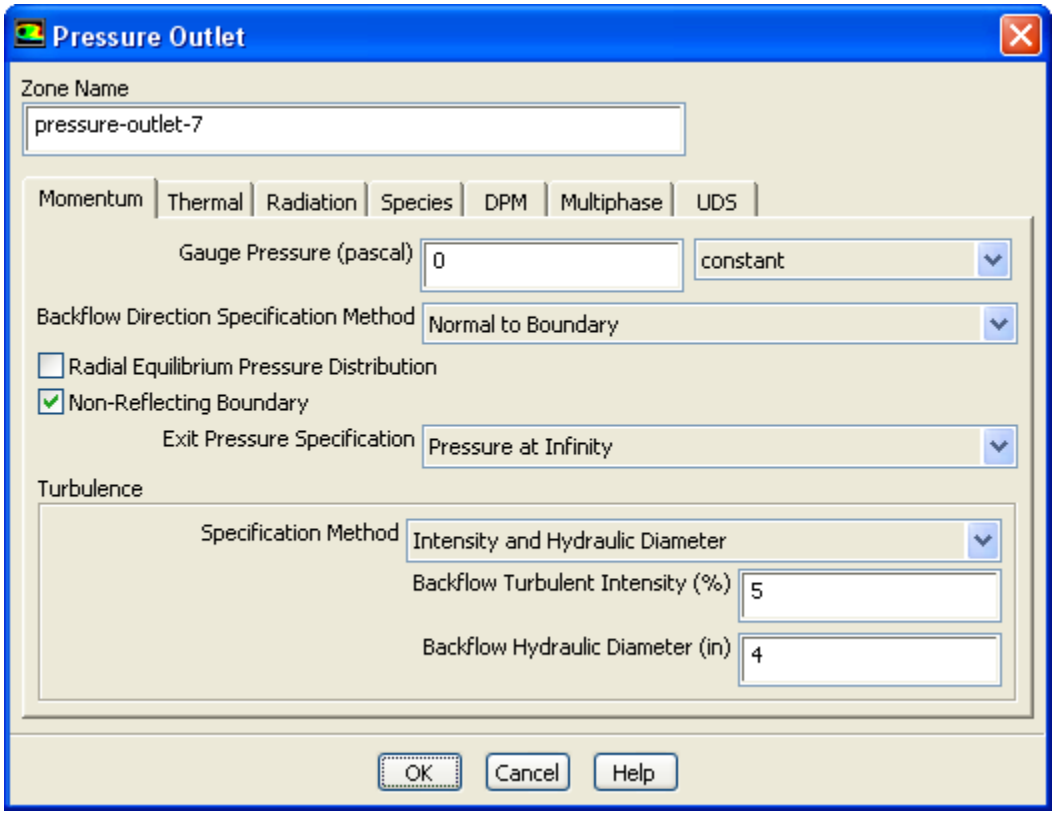
The general NRBC is available for use in the **Pressure Outlet** dialog box when either the density-based explicit or the density-based implicit solvers are activated to solve for compressible flows using the ideal-gas law.

To activate the general NRBC

1. Select **pressure-outlet** from the **Boundary Conditions** task page and click the **Edit...** button.
2. In the **Pressure Outlet** dialog box, enable the **Non-Reflecting Boundary** option.
3. Select one of the two **Exit Pressure Specification** options: **Pressure at Infinity** or **Average Boundary Pressure**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.56 The Pressure Outlet Dialog Box With the Non-Reflecting Boundary Enabled



a. The **Pressure at Infinity** boundary is typically used in unsteady calculations or when the exit pressure value is imposed at infinity. The boundary is designed so that the pressure at the boundary relaxes toward the imposed pressure at infinity. The speed at which this relaxation takes place is controlled by the parameter, `sigma`, which can be adjusted in the TUI:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `general-nrbc` → `set`

   In the `set/` submenu, you can set the `sigma` value. The default value for `sigma` is 0.15.

b. The **Average Boundary Pressure** specification is usually used in steady-state calculations when you want to force the average pressure on the boundary to approach the exit pressure value. The matching of average exit pressure to the imposed average pressure is controlled by the parameter `sigma2` which can be adjusted in the TUI:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `general-nrbc` → `set`

   In the `set/` submenu, you can set the `sigma2` value. The default value for `sigma2` is 5.0.

Important

There is no guarantee that the `sigma2` value of 5.0 will force the average boundary pressure to match the specified exit pressure in all flow situations. In the case where the desired average boundary pressure has not been achieved, the user can intervene to adjust the `sigma2` value so that the desired average pressure on the boundary is approached.

Usually, the solver can operate at higher CFL values without the NRBCs being turned on. Therefore, for steady-state solutions the best practice is to first achieve a good stable solution (not necessarily converged)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

before activating the non-reflecting boundary condition. In many flow situations, the CFL value must be reduced from the normal operation to keep the solution stable. This is particularly true with the density-based implicit solver since the boundary update is done in an explicit manner. A typical CFL value in the density-based implicit solver, with the NRBC activated, is 2.0.

7.5. User-Defined Fan Model

The user-defined fan model in ANSYS FLUENT allows you to periodically regenerate a profile file that can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan.

For example, consider the calculation of the pressure jump across the fan. You can, through the standard interface, input a constant for the pressure jump, specify a polynomial that describes the pressure jump as a function of axial velocity through the fan, or use a profile file that describes the pressure jump as a function of the axial velocity or location at the fan face. If you use a profile file, the same profile will be used consistently throughout the course of the solution. Suppose, however, that you want to change the profile as the flow field develops. This would require a periodic update to the profile file itself, based upon some instructions that you supply. The user-defined fan model is designed to help you do this.

To use this model, you need to generate an executable that reads a fan profile file that is written by ANSYS FLUENT, and writes out a modified one, which ANSYS FLUENT will then read. The source code for this executable can be written in any programming language (Fortran or C, for example). Your program will be called and executed automatically, according to inputs that you supply through the standard interface.

Information about the user-defined fan model is provided in the following sections.

7.5.1. Steps for Using the User-Defined Fan Model
7.5.2. Example of a User-Defined Fan

7.5.1. Steps for Using the User-Defined Fan Model

To make use of the user-defined fan model, follow the steps below.

1. In your model, identify one or more interior faces to represent one or more fan zones.

   Boundary Conditions

2. Input the name of your executable and the instructions for reading and writing profile files in the *User-Defined Fan Model Dialog Box* (p. 2115).

   Define → **User-Defined** → **Fan Model...**

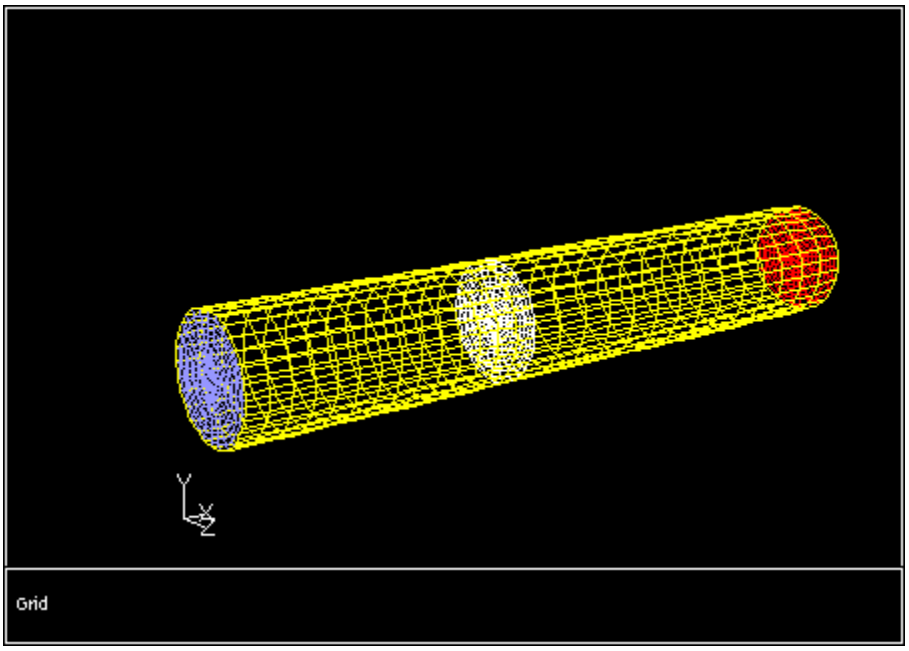
3. Initialize the flow field and the profile files.
4. Enter the fan parameters using the standard *Fan Dialog Box* (p. 1819) (opened from the *Boundary Conditions Task Page* (p. 1812)).
5. Perform the calculation.

7.5.2. Example of a User-Defined Fan

Usage of the user-defined fan model is best demonstrated by an example. With this in mind, consider the domain shown in *Figure 7.57* (p. 360). An inlet supplies air at 10 m/s to a cylindrical region, 1.25 m long and 0.2 m in diameter, surrounded by a symmetry boundary. At the center of the flow domain is a circular fan. A pressure outlet boundary is at the downstream end.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.57 The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain



Solving this problem with the user-defined fan model will cause ANSYS FLUENT to periodically write out a radial profile file with the current solution variables at the fan face. These variables (static pressure, pressure jump, axial, radial, and swirling (tangential) velocity components) will represent averaged quantities over annular sections of the fan. The sizes of the annular regions are determined by the size of the fan and the number of radial points to be used in the profiles.

Once the profile file is written, ANSYS FLUENT will invoke an executable, which will perform the following tasks:

1. Read the profile file containing the current flow conditions at the fan.
2. Perform a calculation to compute new values for the pressure jump, radial velocity, and swirl velocity for the fan.
3. Write a new profile file that contains the results of these calculations.

ANSYS FLUENT will then read the new profile file and continue with the calculation.

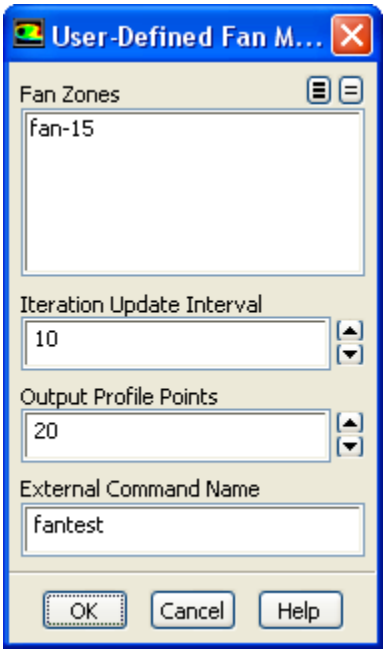
7.5.2.1. Setting the User-Defined Fan Parameters

Specification of the parameters for the user-defined fan begins in the *User-Defined Fan Model Dialog Box* (p. 2115) (*Figure 7.58* (p. 361)).

Define → **User-Defined** → **Fan Model...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.58 The User-Defined Fan Model Dialog Box



In this dialog box, you can select the fan zone(s) on which your executable will operate under **Fan Zones**. In this example, there is only one fan, **fan-8**. If you have multiple fan zones in a simulation, for which you have different profile specifications, you can select them all at this point. Your executable will be able to differentiate between the fan zones because the zone ID for each fan is included in the solution profile file. The executable will be invoked once for each zone, and separate profile files will be written for each.

The executable file will be called on to update the profile file periodically, based on the input for the **Iteration Update Interval**. An input of `10`, as shown in the dialog box, means that the fan executable in this example will act every 10 iterations to modify the profile file.

The number of points in the profile file to be written by ANSYS FLUENT is entered under **Output Profile Points**. This profile file can have the same or a different number of points as the one that is written by the external executable.

Finally, the name of the executable should be entered under **External Command Name**. In the current example, the name of the executable is `fantest`.

Important

If the executable is not located in your working directory, then you must type the complete path to the executable.

7.5.2.2. Sample User-Defined Fan Program

The executable file will be built from the Fortran program, `fantest.f`, which is shown below. You can obtain a copy of this subroutine and the two that it calls (to read and write profile files) by contacting your ANSYS FLUENT technical support engineer.

```
c
c  This program is invoked at intervals by ANSYS FLUENT to
c  read a profile-format file that contains radially
c  averaged data at a fan face, compute new pressure-jump
c  and swirl-velocity components, and write a new profile
```

```
c  file that will subsequently be read by ANSYS FLUENT to
c  update the fan conditions.
c
c  Usage: fantest input_profile output_profile
c

  integer npmax
  parameter (npmax = 900)
  integer inp         ! input: number of profile points
  integer iptype      ! input: profile type (0=radial, 1=point)
  real ir(npmax)      ! input: radial positions
  real ip(npmax)      ! input: pressure
  real idp(npmax)     ! input: pressure-jump
  real iva(npmax)     ! input: axial velocity
  real ivr(npmax)     ! input: radial velocity
  real ivt(npmax)     ! input: tangential velocity
  character*80 zoneid
  integer rfanprof    ! function to read a profile file
  integer status
c
  status = rfanprof(npmax,zoneid,iptype,
 $  inp,ir,ip,idp,iva,ivr,ivt)
  if (status.ne.0) then
    write(*,*) 'error reading input profile file'
  else
    do 10 i = 1, inp
      idp(i) = 200.0 - 10.0*iva(i)
      ivt(i) = 20.0*ir(i)
      ivr(i) = 0.0
 10  continue
  call wfanprof(6,zoneid,iptype,inp,ir,idp,ivr,ivt)
  endif
  stop
  end
```

After the variable declarations, which have comments on the right, the subroutine `rfanprof` is called to read the profile file, and pass the current values of the relevant variables (as defined in the declaration list) to `fantest`. A loop is done on the number of points in the profile to compute new values for:

- The pressure jump across the fan, `idp`, which in this example is a function of the axial velocity, `iva`.
- The swirling or tangential velocity, `ivt`, which in this example is proportional to the radial position, `ir`.
- The radial velocity, `ivr`, which in this example is set to zero.

After the loop, a new profile is written by the subroutine `wfanprof`, shown below. (For more information on profile file formats, see *Profile File Format* (p. 367).)

```
 subroutine wfanprof(unit,zoneid,ptype,n,r,dp,vr,vt)
c
c  writes an ANSYS FLUENT profile file for input by the
c  user fan model
c
  integer unit       ! output unit number
  character*80 zoneid
  integer ptype      ! profile type (0=radial, 1=point)
  integer n          ! number of points
  real r(n)          ! radial position
  real dp(n)         ! pressure jump
  real vr(n)         ! radial velocity
  real vt(n)         ! tangential velocity
  character*6 typenam
  if (ptype.eq.0) then
    typenam = 'radial'
  else
    typenam = 'point'
  endif
  write(unit,*) '((', zoneid(1:index(zoneid,'\0')-1), ' ',
 $  typenam, n, ')'
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    write(unit,*) '(r'
    write(unit,100) r
    write(unit,*) ')'

    write(unit,*) '(pressure-jump'
    write(unit,100) dp
    write(unit,*) ')'

    write(unit,*) '(radial-velocity'
    write(unit,100) vr
    write(unit,*) ')'

    write(unit,*) '(tangential-velocity'
    write(unit,100) vt
    write(unit,*) ')'

   100 format(5(e15.8,1x))
    return
    end
```

This subroutine will write a profile file in either radial or point format, based on your input for the integer `ptype`. (See *Profiles* (p. 366) for more details on the types of profile files that are available.) The names that you use for the various profiles are arbitrary. Once you have initialized the profile files, the names you use in `wfanprof` will appear as profile names in the *Fan Dialog Box* (p. 1819).

7.5.2.3. Initializing the Flow Field and Profile Files

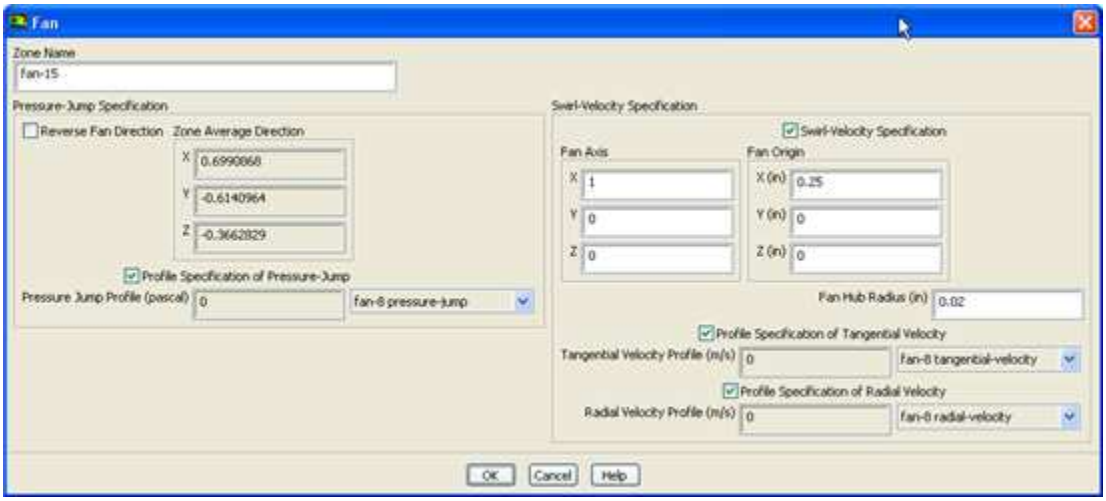
The next step in the setup of the user-defined fan is to initialize (create) the profile files that will be used. To do this, first initialize the flow field with the *Solution Initialization Task Page* (p. 1937) (using the velocity inlet conditions, for example), and then type the command `(update-user-fans)` in the console window. (The parentheses are part of the command, and must be typed in.)

This will create the profile names that are given in the subroutine `wfanprof`.

7.5.2.4. Selecting the Profiles

Once the profile names have been established, you will need to visit the *Fan Dialog Box* (p. 1819) (*Figure 7.59* (p. 363)) to complete the problem setup. (See *Fan Boundary Conditions* (p. 324) for general information on using the **Fan** dialog box.)

Figure 7.59 The Fan Dialog Box



At this time, the **Fan Axis**, **Fan Origin**, and **Fan Hub Radius** can be entered, along with the choice of profiles for the calculation of pressure jump, tangential velocity, and radial velocity. With the profile options enabled, you can select the names of the profiles from the drop-down lists. In the dialog box above, the selected profiles are named **fan-8 pressure-jump**, **fan-8 tangential-velocity**, and **fan-8 radial-velocity**, corresponding to the names that were used in the subroutine `wfanprof`.

7.5.2.5. Performing the Calculation

The solution is now ready to run. As it begins to converge, the report in the console window shows that the profile files are being written and read every 10 iterations:

```
     iter   continuity   x-velocity   y-velocity   z-velocity          k
 !      1   residual normalization factors changed (continuity
        1   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00
 !      2   residual normalization factors changed (continuity
        2   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00   9.4933e-01
        3   6.8870e-01   7.2663e-01   7.3802e-01   7.5822e-01   6.1033e-01
        .       .           .              .           .            .
        .       .           .              .           .            .
        .       .           .              .           .            .
        9   2.1779e-01   9.8139e-02   3.0497e-01   2.9609e-01   2.8612e-01
  Writing "fan-8-out.prof"...
  Done.
  Reading "fan-8-in.prof"...

  Reading profile file...
       10   "fan-8"  radial-profile points, r, pressure-jump,
                          radial-velocity, tangential-velocity.

  Done.
       10   1.7612e-01   7.4618e-02   2.5194e-01   2.4538e-01   2.4569e-01
       11   1.6895e-01   8.3699e-02   2.0316e-01   2.0280e-01   2.1169e-01
        .       .           .              .           .            .
        .       .           .              .           .            .
        .       .           .              .           .            .
```

The file `fan-8-out.prof` is written out by ANSYS FLUENT and read by the executable `fantest`. It contains values for pressure, pressure jump, axial velocity, radial velocity, and tangential velocity at 20 radial locations at the site of the fan. The file `fan-8-in.prof` is generated by `fantest` and contains updated values for pressure jump and radial and tangential velocity only. It is therefore a smaller file than `fan-8-out.prof`. The prefix for these files takes its name from the fan zone with which the profiles are associated. An example of the profile file `fan-8-in.prof` is shown below. This represents the last profile file to be written by `fantest` during the convergence history.

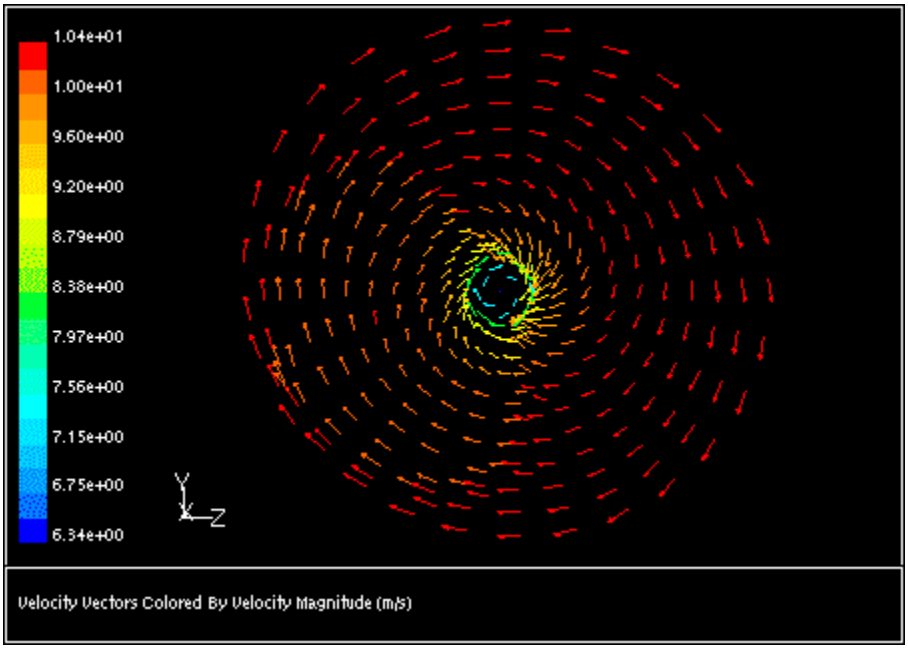
```
((fan-8 radial 10)
(r
0.24295786E-01 0.33130988E-01 0.41966137E-01 0.50801374E-01 0.59636571E-01
0.68471842E-01 0.77307090E-01 0.86142287E-01 0.94963484E-01 0.95353782E-01
)
(pressure-jump
0.10182057E+03 0.98394081E+02 0.97748657E+02 0.97787750E+02 0.97905228E+02
0.98020668E+02 0.98138817E+02 0.98264198E+02 0.98469681E+02 0.98478783E+02
)
(radial-velocity
0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00
0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00
)
(tangential-velocity
0.48591572E+00 0.66261977E+00 0.83932275E+00 0.10160275E+01 0.11927314E+01
0.13694369E+01 0.15461419E+01 0.17228458E+01 0.18992697E+01 0.19070756E+01
)
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.5.2.6. Results

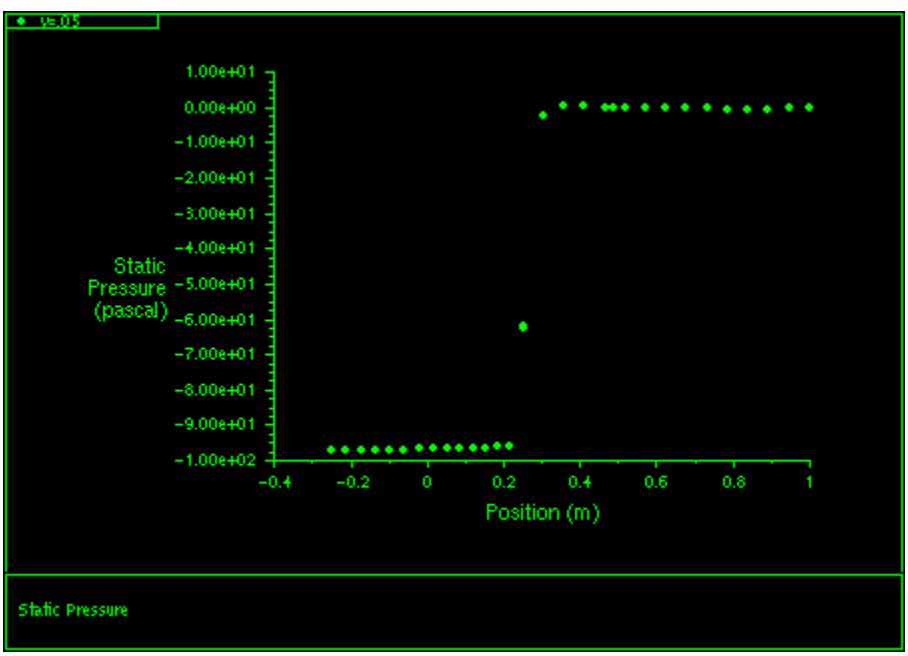
A plot of the transverse velocity components at the site of the fan is shown in *Figure 7.60* (p. 365). As expected, there is no radial component, and the tangential (swirling) component increases with radius.

Figure 7.60 Transverse Velocities at the Site of the Fan



As a final check on the result, an XY plot of the static pressure as a function of x position is shown (*Figure 7.61* (p. 365)). This XY plot is made on a line at y=0.05 m, or at about half the radius of the duct. According to the input file shown above, the pressure jump at the site of the fan should be approximately 97.8 Pa/m. Examination of the figure supports this finding.

Figure 7.61 Static Pressure Jump Across the Fan



7.6. Profiles

Profiles can be boundary conditions, cell zone conditions, and initial conditions for discrete phases. ANSYS FLUENT provides a very flexible profile definition mechanism. This feature allows you to use experimental data, data calculated by an external program, or data written from a previous solution using the *Write Profile Dialog Box* (p. 1811) (as described in *Reading and Writing Profile Files* (p. 70)) as the boundary condition for a variable.

Information about profiles is presented in the following subsections:

7.6.1. Profile Specification Types
7.6.2. Profile File Format
7.6.3. Using Profiles
7.6.4. Reorienting Profiles
7.6.5. Defining Transient Cell Zone and Boundary Conditions

7.6.1. Profile Specification Types

The following is a list of the six types of profiles that can be read into ANSYS FLUENT, as well as information about the interpolation method employed by ANSYS FLUENT for each type.

- Point profiles are specified by an unordered set of n points: $\left(x_i, y_i, v_i\right)$ for 2D problems or $\left(x_i, y_i, z_i, v_i\right)$ for 3D problems, where $1 \le i \le n$. Profiles written using the **Write Profile** dialog box and profiles of experimental data in random order are examples of point profiles.

  ANSYS FLUENT will interpolate the point cloud to obtain values at the boundary faces. The default interpolation method for the unstructured point data is zeroth order. In other words, for each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, to get an accurate specification of an inlet profile using the default interpolation method, your profile file should contain a sufficiently high point density. For information about other available interpolation methods for point profiles, see *Using Profiles* (p. 368).

- Line profiles are specified for 2D problems by an ordered set of n points: $\left(x_i, y_i, v_i\right)$, where $1 \le i \le n$. Zeroth-order interpolation is performed between the points. An example of a line profile is a profile of data obtained from an external program that calculates a boundary-layer profile.

- Mesh profiles are specified for 3D problems by an m by n mesh of points: $\left(x_{ij}, y_{ij}, z_{ij}, v_{ij}\right)$, where $1 \le i \le m$ and $1 \le j \le n$. Zeroth-order interpolation is performed between the points. Examples of mesh profiles are profiles of data from a structured mesh solution and experimental data in a regular array.

- Radial profiles are specified for 2D and 3D problems by an ordered set of n points: (r_i, v_i), where $1 \le i \le n$. The data in a radial profile are a function of radius only. Linear interpolation is performed between the points, which must be sorted in ascending order of the r field. The axis for the cylindrical coordinate system is determined as follows:
  - For 2D problems, it is the z-direction vector through (0,0).
  - For 2D axisymmetric problems, it is the x-direction vector through (0,0).
  - For 3D problems involving a swirling fan, it is the fan axis defined in the *Fan Dialog Box* (p. 1819) (unless you are using local cylindrical coordinates at the boundary, as described below).
  - For 3D problems without a swirling fan, it is the rotation axis of the adjacent fluid zone, as defined in the *Fluid Dialog Box* (p. 1796) (unless you are using local cylindrical coordinates at the boundary, as described below).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - For 3D problems in which you are using local cylindrical coordinates to specify conditions at the boundary, it is the axis of the specified local coordinate system.

- Axial profiles are specified for 3D problems by an ordered set of n points: (z_i, v_i), where $1 \le i \le n$. The data in an axial profile are a function of the axial direction. Linear interpolation is performed between the points, which must be sorted in ascending order of the z field.
- Transient profiles are specified for 2D and 3D profiles by an ordered set of n points: $\left(t_i, v_{0,i}, v_{1,i}, v_{2,i}, \ldots\right)$. Linear interpolations is done between the points which must be sorted in ascending order of the t (time or crank angle) field. Examples of transient profiles are transient cell zone and boundary conditions (see *Defining Transient Cell Zone and Boundary Conditions* (p. 376)) and point properties for particle injections (see *Point Properties for Transient Injections* (p. 1049)).

7.6.2. Profile File Format

The format of the profile files is fairly simple. The file can contain an arbitrary number of profiles. Each profile consists of a header that specifies the profile name, profile type (`point`, `line`, `mesh`, `radial`, or `axial`), and number of defining points, and is followed by an arbitrary number of named "fields". Some of these fields contain the coordinate points and the rest contain boundary data.

Important

All quantities, including coordinate values, must be specified in SI units because ANSYS FLUENT does not perform unit conversion when reading profile files.

Parentheses are used to delimit profiles and the fields within the profiles. Any combination of tabs, spaces, and newlines can be used to separate elements.

Important

In the general format description below, " | " indicates that you should input only one of the items separated by |'s and " . . . " indicates a continuation of the list.

```
((profile1-name point|line|radial n)
 (field1-name a1 a2 ... an)
 (field2-name b1 b2 ... bn)
.
.
.
 (fieldf-name f1 f2 ... fn))

((profile2-name mesh m n)
  (field1-name  a11 a12 ... a1n
                a21 a22 ... a2n
                .
                .
                .
                am1 am2 ... amn)
.
.
.
(fieldf-name   f11  f12 ... f1n
               f21  f22 ... f2n
               .
               .
               .
               fm1 fm2 ... fmn))
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Profile names must have all lowercase letters (e.g., `name`). Uppercase letters in profile names are not acceptable. Each profile of type `point`, `line`, and `mesh` must contain fields with names `x`, `y`, and, for 3D, `z`. Each profile of type `radial` must contain a field with name `r`. Each profile of type `axial` must contain a field with name `z`. The rest of the names are arbitrary, but must be valid Scheme symbols. For compatibility with old-style profile files, if the profile type is missing, `point` is assumed.

7.6.2.1. Example

A typical usage of a profile file is to specify the profile of the boundary layer at an inlet. For a compressible flow calculation, this will be done using profiles of total pressure, k, and ε. For an incompressible flow, it might be preferable to specify the inlet value of streamwise velocity, together with k and ε.

Below is an example of a profile file that does this:

```
((turb-prof point 8)
 (x
     4.00000E+00  4.00000E+00  4.00000E+00  4.00000E+00
     4.00000E+00  4.00000E+00  4.00000E+00  4.00000E+00 )
 (y
     1.06443E-03  3.19485E-03  5.33020E-03  7.47418E-03
     2.90494E-01  3.31222E-01  3.84519E-01  4.57471E-01 )
 (u
     5.47866E+00  6.59870E+00  7.05731E+00  7.40079E+00
     1.01674E+01  1.01656E+01  1.01637E+01  1.01616E+01 )
 (tke
     4.93228E-01  6.19247E-01  5.32680E-01  4.93642E-01
     6.89414E-03  6.89666E-03  6.90015E-03  6.90478E-03 )
 (eps
     1.27713E+02  6.04399E+01  3.31187E+01  2.21535E+01
     9.78365E-03  9.79056E-03  9.80001E-03  9.81265E-03 )
 )
```

7.6.3. Using Profiles

The procedure for using a profile to define a particular cell zone or boundary condition is outlined below.

1. Create a file that contains the desired profile, following the format described in *Profile File Format* (p. 367).
2. Read the profile using the **Read...** button in the *Profiles Dialog Box* (p. 1808) (*Figure 7.62* (p. 370))

   or the **File/Read/Profile...** menu item.

   Cell Zone Conditions → **Profiles...**

   Boundary Conditions → **Profiles...**

   File → **Read** → **Profile...**

   Note that if you use the **Profiles** dialog box to read a file, and a profile in the file has the same name as an existing profile, the old profile will be overwritten.
3. If it is a point profile, you can choose the method of interpolation using the **Profiles** dialog box (*Figure 7.62* (p. 370)):

   Cell Zone Conditions → **Profiles...**

   Boundary Conditions → **Profiles...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the point profile in the **Profile** selection list. Then select one of the three choices in the **Interpolation Method** list and click the **Apply** button. The three choices include:

- **Constant**

  This method is zeroth-order interpolation. For each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, the accuracy of the interpolated profile will be affected by the density of the data points in your profile file. This is the default interpolation method for point profiles.

- **Inverse Distance**

  This method assigns a value to each cell face at the boundary based on weighted contributions from the values in the profile file. The weighting factor is inversely proportional to the distance between the profile point and the cell face center.

- **Least Squares**

  This method assigns values to the cell faces at the boundary through a first-order interpolation method that tries to minimizes the sum of the squares of the offsets (residuals) between the profile data points and the cell face centers. The least squares solution is found using Singular Value Decomposition (SVD).

For information about the interpolation methods employed for other profile types (i.e., line, mesh, radial, or axial profiles), see *Profile Specification Types* (p. 366).

4. In the boundary conditions dialog boxes (e.g., the **Velocity Inlet** and **Pressure Inlet** dialog boxes), the fields defined in the profile file (and those defined in any other profile file that you have read in) will appear in the drop-down list to the right of or below each parameter for which profile specification is allowed. To use a particular profile, select it in the appropriate list.
5. Initialize the solution to interpolate the profile.

Important

Profiles *cannot* be used to define volumetric source terms. If you want to define a non-constant source term, you will need to use a user-defined function.

For more information on UDFs, refer to the UDF Manual.

7.6.3.1. Checking and Deleting Profiles

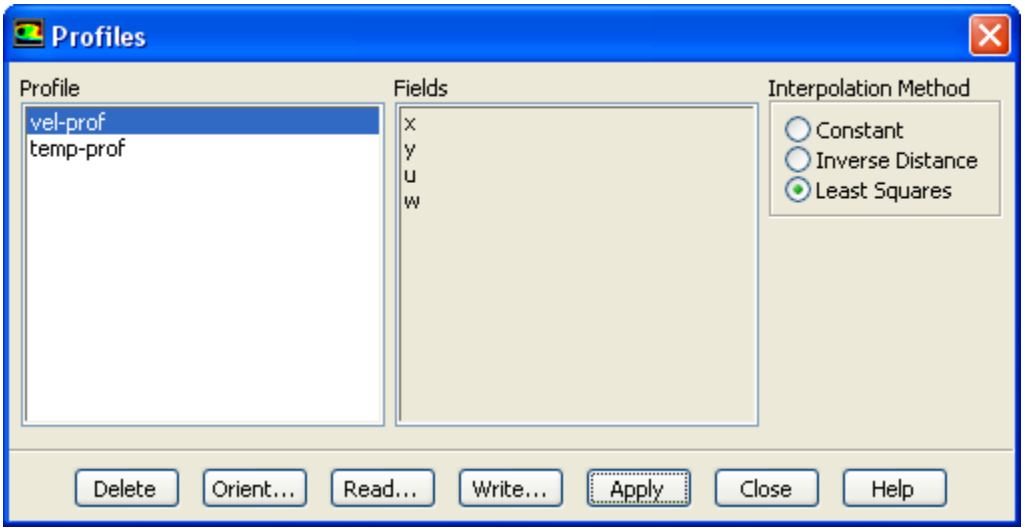
Each profile file contains one or more profiles, and each profile has one or more fields defined in it. Once you have read in a profile file, you can check which fields are defined in each profile, and you can also delete a particular profile. These tasks are accomplished in the *Profiles Dialog Box* (p. 1808) (*Figure 7.62* (p. 370)).

Cell Zone Conditions → **Profiles...**

Boundary Conditions → **Profiles...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.62 The Profiles Dialog Box



To check which fields are defined in a particular profile, select the profile name in the **Profile** list. The available fields in that file will be displayed in the **Fields** list. In *Figure 7.62* (p. 370), the profile fields from the profile file of *Example* (p. 368) are shown.

To delete a profile, select it in the **Profile** list and click the **Delete** button. When a profile is deleted, all fields defined in it will be removed from the **Fields** list.

7.6.3.2. Viewing Profile Data

The **Plots** task page options allow you to generate XY plots of data related to profiles. You can plot the original data points from the profile file you have read into ANSYS FLUENT, or you can plot the values assigned to the cell faces on the boundary after the profile file has been interpolated. See *XY Plots of Profiles* (p. 1474) for the steps to generate these plots.

You have the additional option of viewing the parameters) using the **Plot** or the **Contours** options. Note that these display options do not allow you to plot the actual values of the cell faces (as is done with the **Interpolated Data** option), because they interpolate the values stored in the adjacent cells. To view the boundary condition parameters you must first read in the profile, save a boundary condition with a profile field selected as a parameter, and initialize the flow solution. Then you can you can view the surface data as follows:

- For 2D calculations, open the **Solution XY Plot** dialog box. Select the appropriate boundary zone in the **Surfaces** list, the variable of interest in the **Y Axis Function** drop-down list, and the desired **Plot Direction**. Ensure that the **Node Values** check button is turned on, and then click **Plot**. You should then see the profile plotted. If the data plotted does not agree with your specified profile, this means that there is an error in the profile file.
- For 3D calculations, use the **Contours** dialog box to display contours on the appropriate boundary zone surface. The **Node Values** check button must be turned on in order for you to view the profile data. If the data shown in the contour plot does not agree with your specified profile, this means that there is an error in the profile file.

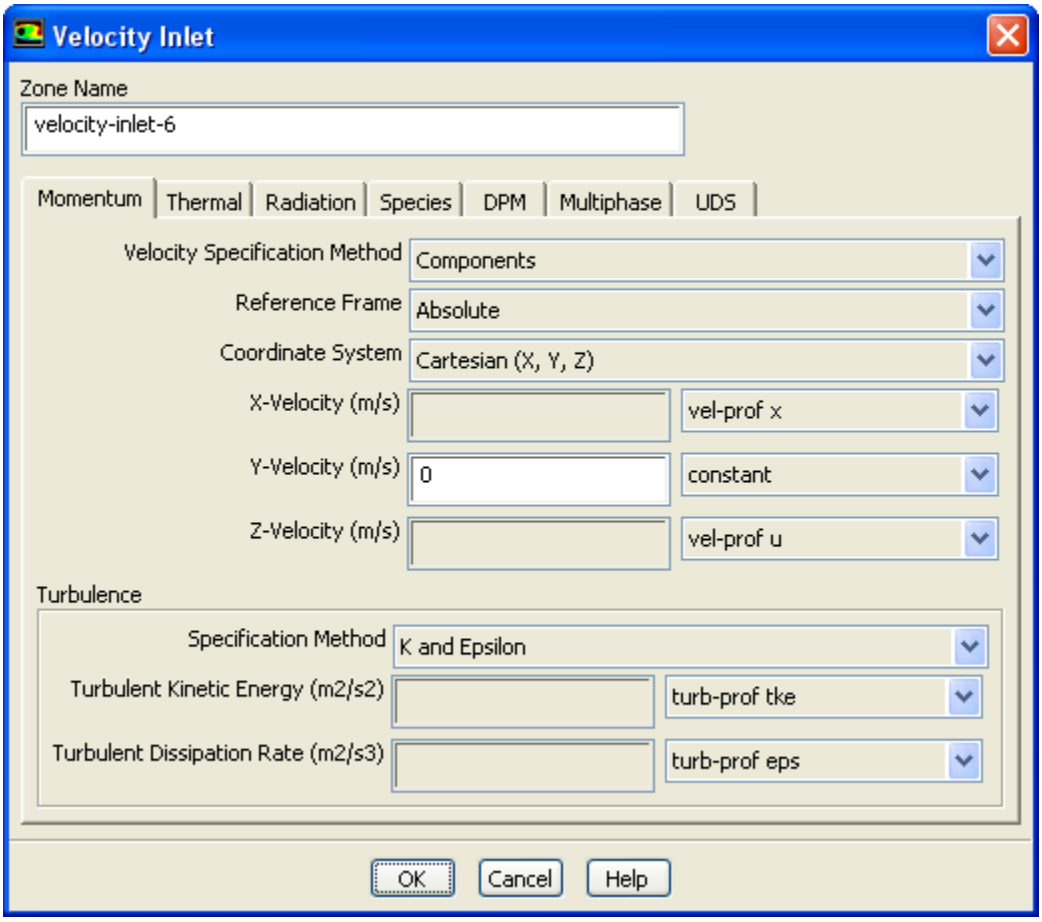
7.6.3.3. Example

For the example given in *Example* (p. 368), the profiles are used for inlet values of x velocity, turbulent kinetic energy, and turbulent kinetic energy dissipation rate, as illustrated in *Figure 7.63* (p. 371). (The y velocity is

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set to a constant value of zero, since it is assumed negligible. However, a profile of y velocity could also be used.)

Figure 7.63 Example of Using Profiles as Boundary Conditions



7.6.4. Reorienting Profiles

For 3D cases only, ANSYS FLUENT allows you to change the orientation of an existing profile so that it can be used at a boundary positioned arbitrarily in space. This allows you, for example, to take experimental data for an inlet with one orientation and apply it to an inlet in your model that has a different spatial orientation. Note that ANSYS FLUENT assumes that the profile and the boundary are planar.

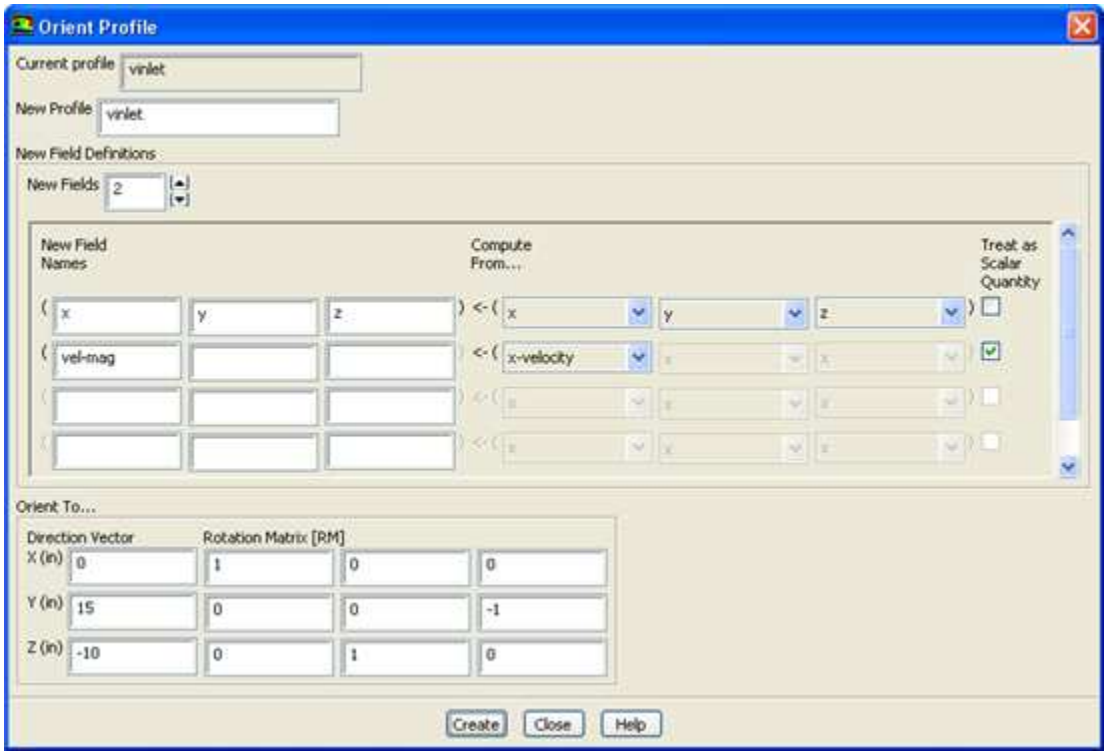
7.6.4.1. Steps for Changing the Profile Orientation

The procedure for orienting the profile data in the principal directions of a boundary is outlined below:

1. Define and read the profile as described in *Using Profiles* (p. 368).
2. In the *Profiles Dialog Box* (p. 1808), select the profile in the **Profile** list, and then click the **Orient...** button. This will open the *Orient Profile Dialog Box* (p. 1810) (*Figure 7.64* (p. 372)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.64 The Orient Profile Dialog Box



3. In the **Orient Profile** dialog box, enter the name of the new profile you want to create in the **New Profile** box.
4. Specify the number of fields you want to create using the up/down arrows next to the **New Fields** box. The number of new fields is equal to the number of vectors and scalars to be defined plus 1 (for the coordinates).
5. Define the coordinate field.
   a. Enter the names of the three coordinates (x, y, z) in the first row under **New Field Names**.

   Important

   Ensure that the coordinates are named x, y, and z only. Do not use any other names or upper case letters in this field.

   b. Select the appropriate local coordinate fields for x, y, and z from the drop-down lists under **Compute From...**. (A selection of **0** indicates that the coordinate does not exist in the original profile; i.e., the original profile was defined in 2D.)
6. Define the vector fields in the new profile.
   a. Enter the names of the 3 components in the directions of the coordinate axes of the boundary under **New Field Names**.

   Important

   Do not use upper case letters in these fields.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the names of the 3 components of the vector in the local x, y, and z directions of the profile from the drop-down lists under **Compute From...**.

7. Define the scalar fields in the new profile.

a. Enter the name of the scalar in the first column under **New Field Names**.

Important

Do not use upper case letters in these fields.

b. Click the button under **Treat as Scalar Quantity** in the same row.

c. Select the name of the scalar in the corresponding drop-down list under **Compute From...**.

8. Under **Orient To...**, specify the rotational matrix RM under the **Rotation Matrix [RM]**. The rotational matrix used here is based on Euler angles (γ, β, and α) that define an orthogonal system $x'y'z'$ as the result of the three successive rotations from the original system xyz. In other words,

$$\begin{bmatrix} x' \\ y' \\ z' \end{bmatrix} = [RM] \begin{bmatrix} x \\ y \\ z \end{bmatrix} \tag{7–164}$$

$$RM = [C][B][A] \tag{7–165}$$

where C, B, and A are the successive rotations around the z, y, and x axes, respectively.

Rotation around the z axis:

$$C = \begin{bmatrix} \cos\gamma & -\sin\gamma & 0 \\ \sin\gamma & \cos\gamma & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{7–166}$$

Rotation around the y axis:

$$B = \begin{bmatrix} \cos\beta & 0 & \sin\beta \\ 0 & 1 & 0 \\ -\sin\beta & 0 & \cos\beta \end{bmatrix} \tag{7–167}$$

Rotation around the x axis:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$A = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\alpha & -\sin\alpha \\ 0 & \sin\alpha & \cos\alpha \end{bmatrix} \tag{7–168}$$

9. Under **Orient To...**, specify the **Direction Vector**. The **Direction Vector** is the vector that translates a profile to the new position, and is defined between the centers of the profile fields.

Important

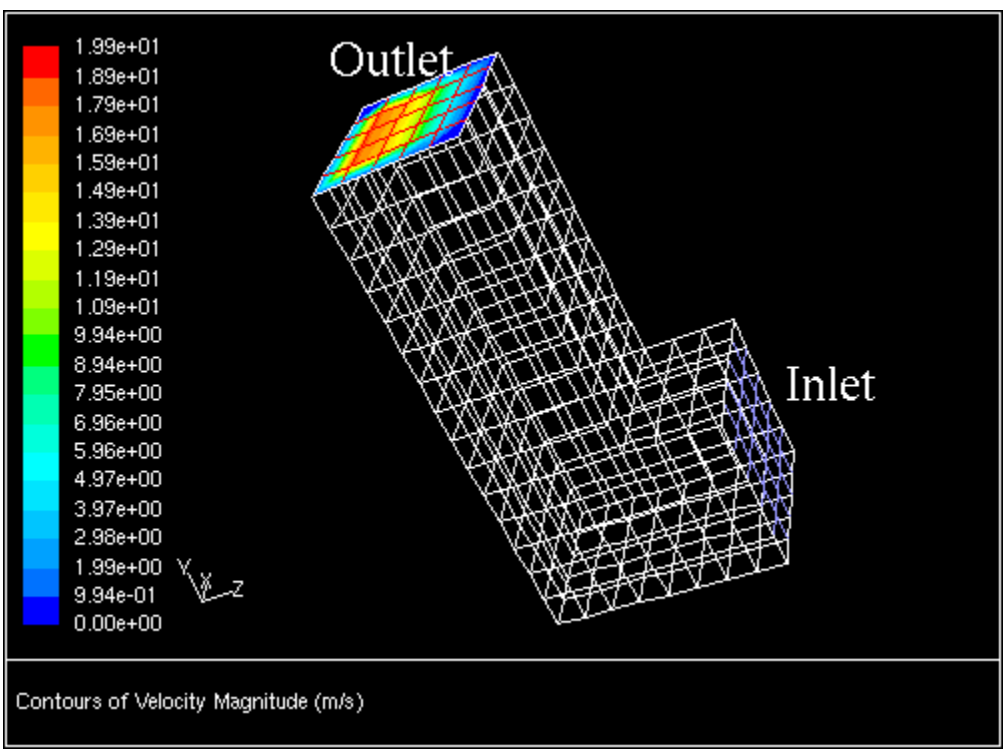
Note that depending on your case, it may be necessary to perform only a rotation, only a translation, or a combination of a translation and a rotation.

10. Click the **Create** button in the **Orient Profile** dialog box, and your new profile will be created. Its name, which you entered in the **New Profile** box, will now appear in the **Profiles** dialog box and will be available for use at the desired boundary.

7.6.4.2. Profile Orienting Example

Consider the domain with a square inlet and outlet, shown in *Figure 7.65* (p. 374). A scalar profile at the outlet is written out to a profile file. The purpose of this example is to impose this outlet profile on the inlet boundary via a90 $^{\circ}$ rotation about the x axis. However, the rotation will locate the profile away from the inlet boundary. To align the profile to the inlet boundary, a translation via a directional vector needs to be performed.

Figure 7.65 Scalar Profile at the Outlet



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The problem is shown schematically in *Figure 7.66* (p. 376). Φ_{out} is the scalar profile of the outlet. Φ'_{out} is the image of the Φ_{out} rotated 90 $^\circ$ around the x axis. In this example, since $\gamma = \beta = 0$, then $C = B = I$, where I is the identity matrix, and the rotation matrix is

$$RM = [C][B][A] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos 90^\circ & -\sin 90^\circ \\ 0 & \sin 90^\circ & \cos 90^\circ \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & -1 \\ 0 & 1 & 0 \end{bmatrix} \quad \textbf{(7–169)}$$

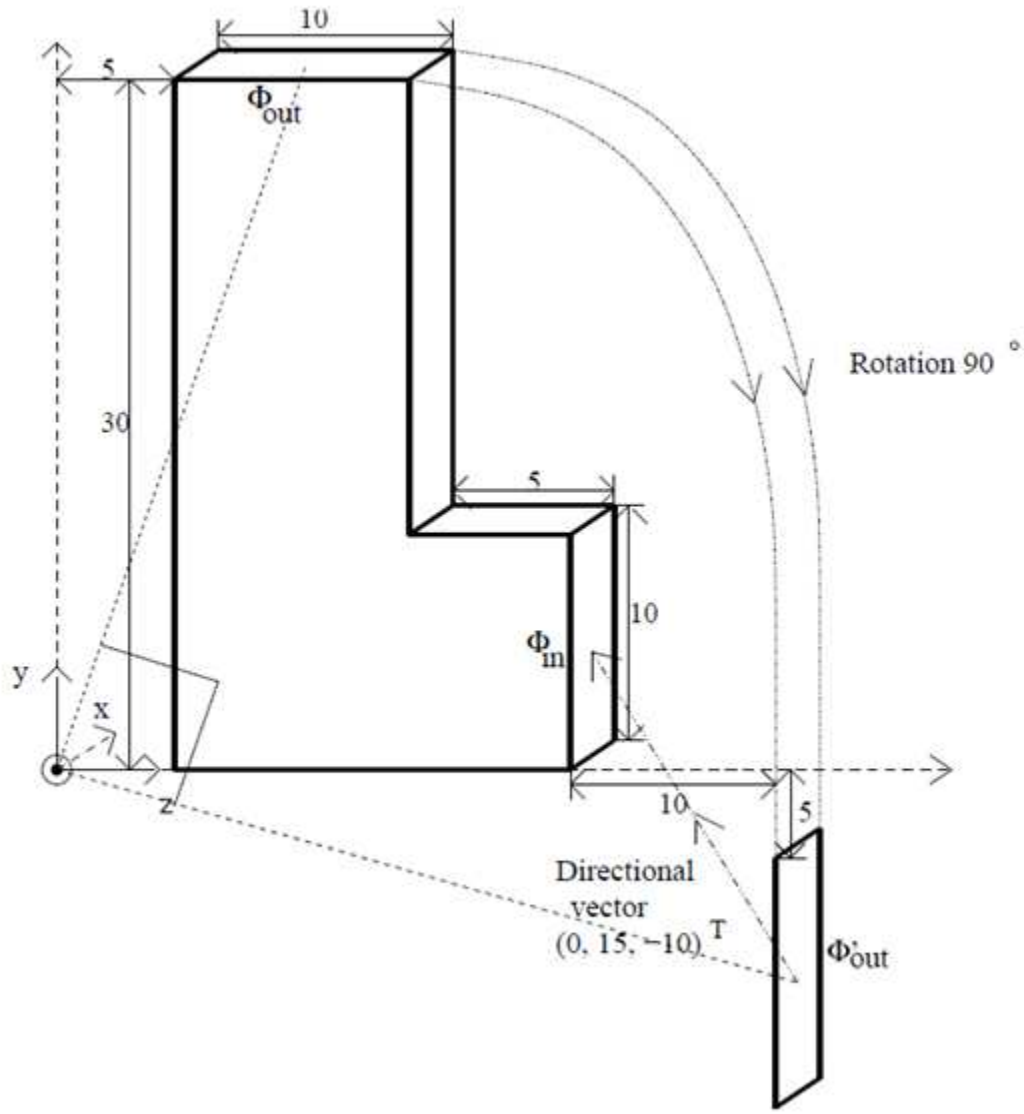
To overlay the outlet profile on the inlet boundary, a translation will be performed.

To overlay the outlet profile on the inlet boundary, a translation will be performed. The directional vector is the vector that translates Φ'_{out} to Φ_{in}. In this example, the directional vector is $(0, 15, -10)^T$. The appropriate inputs for the **Orient Profile** dialog box are shown in *Figure 7.64* (p. 372).

Note that if the profile being imposed on the inlet boundary was due to a rotation of -90 $^\circ$ about the x axis, then the rotational matrix RM must be found for $\gamma = \beta = 0$ and $\alpha = -90^\circ$, and a new directional vector must be found to align the profile to the boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.66 Problem Specification



7.6.5. Defining Transient Cell Zone and Boundary Conditions

There are two ways you can specify transient cell zone and boundary conditions:

- transient profile with a format similar to the standard profiles described in *Profiles* (p. 366)
- transient profile in a tabular format

Important

For both methods, the cell zone or boundary condition will vary only in time; it must be spatially uniform. However, if the in-cylinder model is activated (*In-Cylinder Settings* (p. 587)), then you have the option to use the crank angle instead of time. Crank angles can be included in transient tables as well as transient profiles, in a similar fashion to time. Examples of transient profiles and transient tables in crank angle can be found in the sections that follow.

For information about boundary profiles, please refer to *Reading and Writing Profile Files* (p. 70).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.5.1. Standard Transient Profiles

The format of the standard transient profile file (based on the profiles described in *Profiles* (p. 366)) is

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

The profile name as well as the field names have to be shorter than 64 characters. One of the `field_name`s should be used for the `time` field, and the `time` field section *must* be in ascending order. n is the number of entries per field. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 3 0)
(time
1
2
3
)
(u
10
20
30
)
)
```

This example demonstrates the use of crank angle in a transient profile

```
((example transient 3 1)
(angle
0.000000e+00 1.800000e+02 3.600000e+02)
(temperature
3.000000e+02 5.000000e+02 3.000000e+02)
)
```

Important

All quantities, including coordinate values, must be specified in SI units because ANSYS FLUENT does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (e.g., `name`). Uppercase letters in profile names are not acceptable.

You can read this file into ANSYS FLUENT using the *Profiles Dialog Box* (p. 1808) or the **File/Read/Profile...** menu item.

Cell Zone Conditions → **Profiles...**

Boundary Conditions → **Profiles...**

File → **Read** → **Profile...**

See *Using Profiles* (p. 368) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.5.2. Tabular Transient Profiles

The format of the tabular transient profile file is

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

The first field name (e.g. `field-name-1`) should be used for the `time` field, and the `time` field section, which represents the flow time, *must* be in ascending order. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
sampletabprofile 2 3 0
time u
1 10
2 20
3 30
```

This file defines the same transient profile as the standard profile example above.

If the periodicity is set to `1`, then `n_data` must be the number that closes one period.

An example is shown below:

```
periodtabprofile 2 4 1
time u
0 10
1 20
2 30
3 10
```

The following example uses crank angle instead of time:

```
example 2 3 1
angle temperature
0  300
180 500
360 300
```

Important

All quantities, including coordinate values, must be specified in SI units because ANSYS FLUENT does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (e.g., `name`). Uppercase letters in profile names are not acceptable. When choosing the field names, spaces or parentheses should not be included.

You can read this file into ANSYS FLUENT using the `read-transient-table`

text command.

`file` → `read-transient-table`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After reading the table into ANSYS FLUENT, the profile will be listed in the *Profiles Dialog Box* (p. 1808) and can be used in the same way as a boundary profile. See *Using Profiles* (p. 368) for details.

7.7. Coupling Boundary Conditions with GT-Power

GT-Power users can define time-dependent boundary conditions in ANSYS FLUENT based on information from GT-Power. During the ANSYS FLUENT simulation, ANSYS FLUENT and GT-Power are coupled together and information about the boundary conditions at each time step is transferred between them.

7.7.1. Requirements and Restrictions
7.7.2. User Inputs

7.7.1. Requirements and Restrictions

Note the following requirements and restrictions for the GT-Power coupling:

- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using GT-Power must be a flow boundary of one of the following types:
  - velocity inlet
  - mass flow inlet
  - pressure inlet
  - pressure outlet

  Also, a maximum of 20 boundary zones can be coupled to GT-Power.
- If a mass flow inlet or pressure inlet is coupled to GT-Power, you must select **Normal to Boundary** as the **Direction Specification Method** in the **Mass-Flow Inlet** or **Pressure Inlet** dialog box. For a velocity inlet, you must select **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in the **Velocity Inlet** dialog box.
- The mass flow specification method in the**Mass-Flow Inlet** boundary has to always be **Mass Flux** and not **Mass Flow Rate** when coupling with GTPower.
- Boundary conditions for the following variables can be obtained from GT-Power:
  - velocity
  - temperature
  - pressure
  - density
  - species mass fractions
  - k and ε (Note that it is recommended that you define these conditions in ANSYS FLUENT yourself, rather than using the data provided by GT-Power, since the GT-Power values are based on a 1D model.)
- Make sure that the material properties you set in ANSYS FLUENT are the same as those used in GT-Power, so that the boundary conditions will be valid for your coupled simulation.
- If your model includes species, make sure that the name of each species in GT-Power corresponds to the **Chemical Formula** for that species material in the **Materials** dialog box. Also, recall that ANSYS FLUENT can handle a maximum of 50 species.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can install the GT-Power libraries in a directory other than the default location. If the GT-Power libraries are loaded into a non-default location, you need to set the following environment variables:
  - `ANSYS FLUENT_GTIHOME`- the GTI installation directory where GT-Power is installed
  - `ANSYS FLUENT_GTIVERSION`-the current version of the GTI installation

Important

GTI is not backwards compatible.

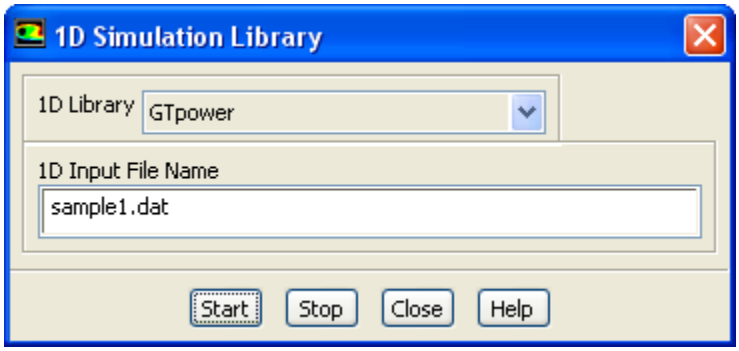
7.7.2. User Inputs

The procedure for setting up the GT-Power coupling in ANSYS FLUENT is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types (but *not* the actual boundary conditions), noting the requirements and restrictions listed in *Requirements and Restrictions* (p. 379).
2. Specify the location of the GT-Power data and have ANSYS FLUENT use them to generate user-defined functions for the relevant boundary conditions (using the *1D Simulation Library Dialog Box* (p. 2116), shown in *Figure 7.67* (p. 380)).

   Define → **User-Defined** → **1D Coupling...**

   Figure 7.67 The 1D Simulation Library Dialog Box

   a. Select **GTpower** in the **1D Library** drop-down list.

   b. Specify the name of the GT-Power input file in the **1D Input File Name** field.

   c. Click the **Start** button.

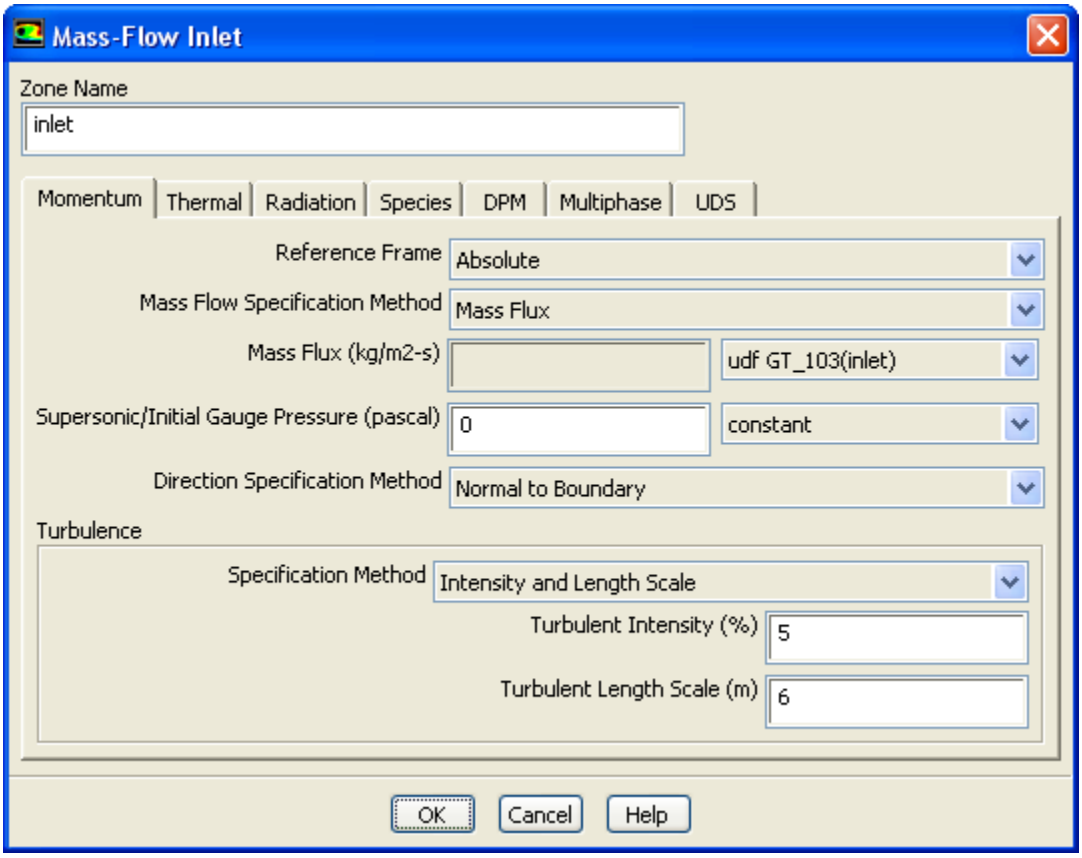
   When you click **Start**, GT-Power will start up and ANSYS FLUENT user-defined functions for each boundary in the input file will be generated.
3. Set boundary conditions for all zones. For flow boundaries for which you are using GT-Power data, select the appropriate UDFs as the conditions.

   Important

   Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in *Figure 7.68* (p. 381)); this UDF contains all of the conditions at that boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.68 Using GT-Power Data for Boundary Conditions



4. If you plan to continue the simulation at a later time, starting from the final data file of the current simulation, specify how often you want to have the case and data files saved automatically.

   Calculation Activities (Autosave Case/Data) → Edit...

   To use a GT-Power restart file to restart an ANSYS FLUENT calculation, you must edit the GT-Power input data file. See the GT-Power User's Guide for instructions.

5. Continue the problem setup and calculate a solution in the usual manner.

7.8. Coupling Boundary Conditions with WAVE

WAVE users can define time-dependent boundary conditions in ANSYS FLUENT based on information from WAVE. During the ANSYS FLUENT simulation, ANSYS FLUENT and WAVE are coupled together and information about the boundary conditions at each time step is transferred between them.

7.8.1. Requirements and Restrictions
7.8.2. User Inputs

7.8.1. Requirements and Restrictions

Note the following requirements and restrictions for the WAVE coupling:

- WAVE needs to be installed and licensed.
- There are always five species that must be modeled in ANSYS FLUENT just as they are defined in WAVE (F1, F2, F3, F4, and F5). It is recommended that realistic material properties be assigned to each of the five species.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using WAVE must be a flow boundary of one of the following types:
  - velocity inlet
  - mass flow inlet
  - pressure inlet
  - pressure outlet

  Also, a maximum of 20 boundary zones can be coupled to WAVE.
- If a mass flow inlet or pressure inlet is coupled to WAVE, you must select **Normal to Boundary** as the **Direction Specification Method** in the **Mass-Flow Inlet** or **Pressure Inlet** Dialog Boc. For a velocity inlet, you must select **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in the **Velocity Inlet** Dialog Box.
- Boundary conditions for the following variables can be obtained from WAVE:
  - velocity
  - temperature
  - pressure
  - density
  - species mass fractions
  - k and ε (Note that you are required to define these conditions in ANSYS FLUENT yourself, since WAVE does not calculate them.)
- Make sure that the material properties you set in ANSYS FLUENT are the same as those used in WAVE, so that the boundary conditions will be valid for your coupled simulation.
- If your model includes species, make sure that the name of each species in WAVE corresponds to the **Chemical Formula** for that species material in the **Create/Edit Materials** dialog Box. Also, recall that ANSYS FLUENT can handle a maximum of 50 species.

7.8.2. User Inputs

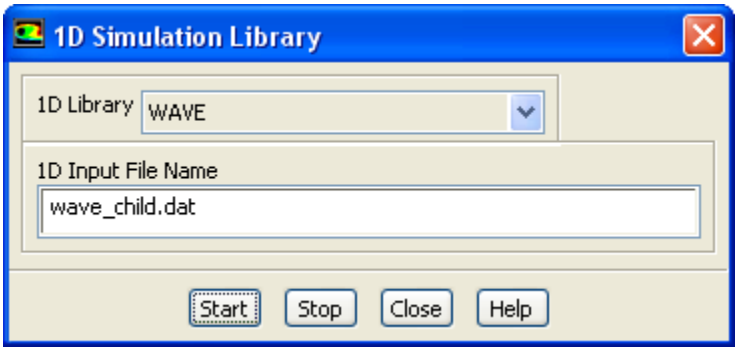
The procedure for setting up the WAVE coupling in ANSYS FLUENT is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types.
2. Specify the location of the WAVE data and have ANSYS FLUENT use them to generate user-defined functions for the relevant boundary conditions (using the *1D Simulation Library Dialog Box* (p. 2116), shown in *Figure 7.69* (p. 383)).

   Define $\rightarrow$ **User-Defined** $\rightarrow$ **1D Coupling...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.69 The 1D Simulation Library Dialog Box with WAVE Selected



a. Select **WAVE** in the **1D Library** drop-down list.

b. Specify the name of the WAVE input file in the **1D Input File Name** field.

c. Click the **Start** button.

When you click **Start**, WAVE will start up and ANSYS FLUENT user-defined functions for each boundary in the input file will be generated.

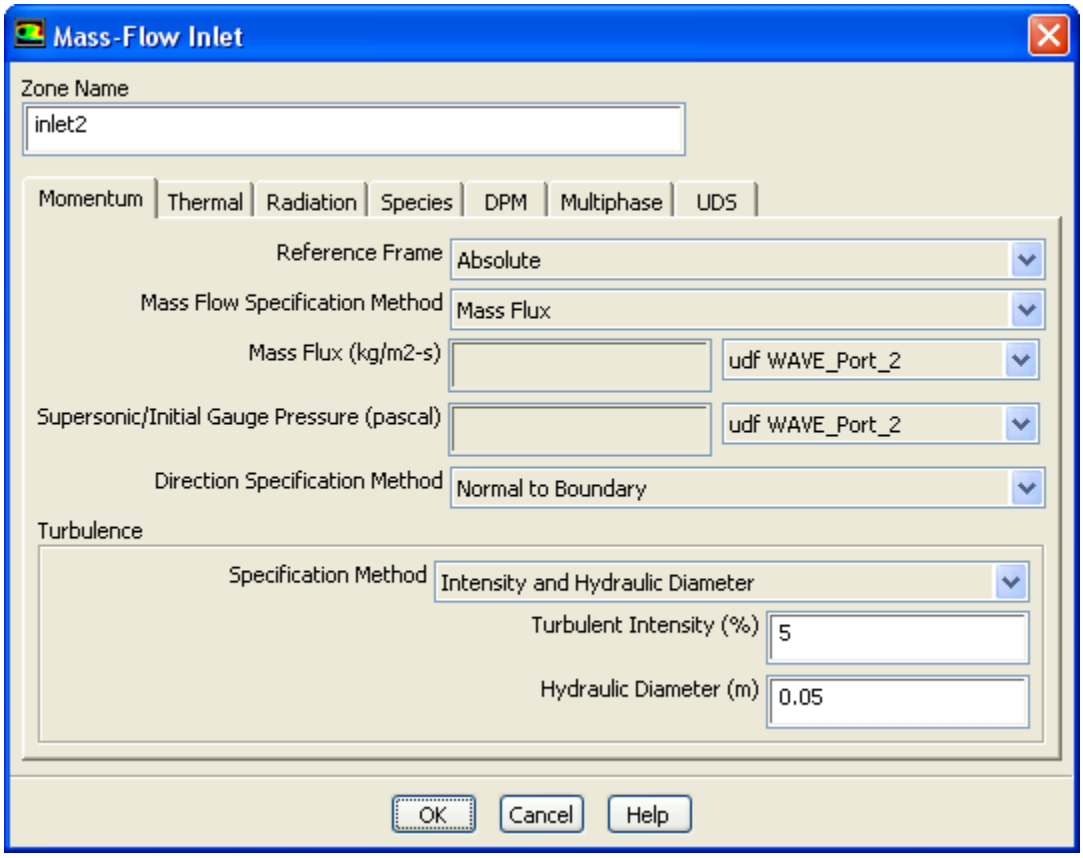
3. Set boundary conditions for all zones. For flow boundaries for which you are using WAVE data, select the appropriate UDFs as the conditions.

Important

Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in *Figure 7.70* (p. 384)); this UDF contains all of the conditions at that boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.70 Using WAVE Data for Boundary Conditions



4. If you plan to continue the simulation at a later time, restarting from the final data file of the current simulation, you need to instruct both ANSYS FLUENT and WAVE how often that you want to automatically save your data. You should instruct ANSYS FLUENT to automatically save case and data files at specified intervals using the autosave feature.

   Calculation Activities (Autosave Case/Data) → Edit...

   In addition, you should instruct WAVE as to how often it should generate its own restart files. See the WAVE User's Guide for instructions on this feature.

   Important

   To use the restart feature, the time interval for writing data files must be set to the same value in both ANSYS FLUENT and WAVE. For example, if ANSYS FLUENT has set the autosave feature to 100, then WAVE must also set the restart file write frequency to 100 as well.

5. Continue the problem setup and calculate a solution in the usual manner.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: Physical Properties

This chapter describes how to define materials, the physical equations used to compute material properties, and the methods you can use for each property input. Each property is described in detail in the following sections. If you are using one of the general multiphase models (VOF, mixture, or Eulerian), see *Defining the Phases* (p. 1098) for information about how to define the individual phases and their material properties.

8.1. Defining Materials
8.2. Defining Properties Using Temperature-Dependent Functions
8.3. Density
8.4. Viscosity
8.5. Thermal Conductivity
8.6. User-Defined Scalar (UDS) Diffusivity
8.7. Specific Heat Capacity
8.8. Radiation Properties
8.9. Mass Diffusion Coefficients
8.10. Standard State Enthalpies
8.11. Standard State Entropies
8.12. Molecular Heat Transfer Coefficient
8.13. Kinetic Theory Parameters
8.14. Operating Pressure
8.15. Reference Pressure Location
8.16. Real Gas Models

8.1. Defining Materials

An important step in the setup of the model is to define the materials and their physical properties. Material properties are defined in the *Materials Task Page* (p. 1735), where you can enter values for the properties that are relevant to the problem scope you have defined in the *Models Task Page* (p. 1639). These properties may include the following:

- density and/or molecular weights
- viscosity
- heat capacity
- thermal conductivity
- UDS diffusion coefficients
- mass diffusion coefficients
- standard state enthalpies
- kinetic theory parameters

Properties may be temperature-dependent and/or composition-dependent, with temperature dependence based on a polynomial, piecewise-linear, or piecewise-polynomial function and individual component properties either defined by you or computed via kinetic theory.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Materials Task Page* (p. 1735) will show the properties that need to be defined for the active physical models. If any property you define requires the energy equation to be solved (e.g., ideal gas law for density, temperature-dependent profile for viscosity), ANSYS FLUENT will automatically activate the energy equation. Then you have to define the thermal boundary conditions and other parameters yourself.

For additional information, please see the following sections:

8.1.1. Physical Properties for Solid Materials
8.1.2. Material Types and Databases
8.1.3. Using the Materials Task Page
8.1.4. Using a User-Defined Materials Database

8.1.1. Physical Properties for Solid Materials

For solid materials, only density, thermal conductivity, and heat capacity are defined. If you are modeling semi-transparent media, case radiation properties are also defined. You can specify a constant value, a temperature-dependent function, or a user-defined function for thermal conductivity; a constant value or temperature-dependent function for heat capacity; and a constant value for density.

If you are using the pressure-based solver, density and heat capacity for a solid material are not required unless you are modeling transient flow or moving solid zones. Heat capacity will appear in the list of solid properties for steady flows as well. The value will be used just for postprocessing enthalpy; not in the calculation.

8.1.2. Material Types and Databases

In ANSYS FLUENT, you can define six types of materials: fluids, solids, mixtures, combusting-particles, droplet-particles, and inert-particles. Physical properties of fluids and solids are associated with named materials are then assigned as boundary conditions for zones.

When you model species transport, define a mixture material, consisting of the various species involved in the problem. Properties will be defined for the mixture, as well as for the constituent species, which are fluid materials. The mixture material concept is discussed in detail in *Mixture Materials* (p. 800). Combusting-particles, droplet-particles, and inert-particles are available for the discrete-phase model, as described in *The Concept of Discrete-Phase Materials* (p. 1057).

ANSYS FLUENT provides a built-in global database of approximately 675 predefined materials from this global (site-wide) database and use the default properties or define new materials database is located in the following file:

path `/Fluent.Inc/fluent13.0/cortex/lib/propdb.scm`

where *path* is the directory in which you have installed ANSYS FLUENT.

In addition to using the ANSYS FLUENT materials, and use it to define the materials in your problem setup. See *Using a User-Defined Materials Database* (p. 391) for information about creating and using user-defined custom material databases.

Important

All the materials list will be saved in the case file (when you write one). The materials specified by you will be available to you if you read this case file into a new solver session.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

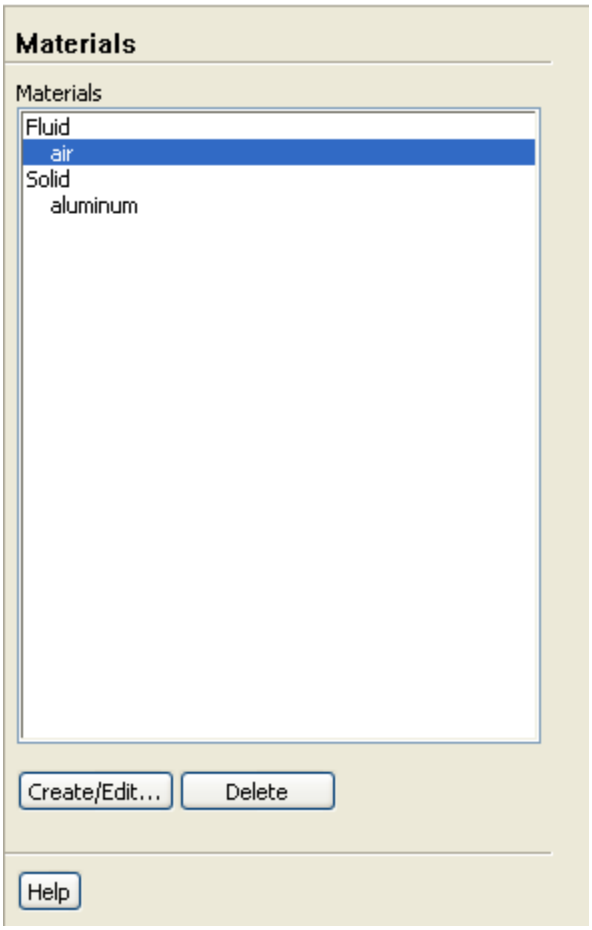
8.1.3. Using the Materials Task Page

The *Materials Task Page* (p. 1735) (*Figure 8.1* (p. 387)) allows you to define the materials and their properties in your problem setup using either the **Fluent Database** or a **User-Defined Database**. It allows you to copy materials, and modify material properties.

These generic functions are described in this section. The inputs for temperature-dependent properties are explained in *Defining Properties Using Temperature-Dependent Functions* (p. 398). The specific inputs for each material property are discussed in the remaining sections of this chapter.

Materials

Figure 8.1 The Materials Task Page



By default, your local materials list will include a single fluid material (air) and a single solid material (aluminum). If the fluid involved in your problem is air, you can use the default properties for air or modify the properties. If the fluid in your problem is water, you can either copy water from the ANSYS FLUENT database or create a new "water" material from scratch. If you copy water from the database, you can still make modifications to the properties of your local copy of water. The editing or creating of material properties is done in the *Create/Edit Materials Dialog Box* (p. 1736). To display the *Create/Edit Materials Dialog Box* (p. 1736), select the **Create/Edit...** button in the *Materials Task Page* (p. 1735). See the following sections for detailed information on how to change material properties.

Mixture materials will not exist in your local list unless you have enabled species transport (see *Modeling Species Transport and Finite-Rate Chemistry* (p. 799)). Similarly, inert, droplet, and combusting particle materials

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will not be available unless you have created a discrete phase injection of these particle types (see *Modeling Discrete Phase* (p. 1007)). When a mixture material is copied from the database, all of its constituent fluid materials (species) will automatically be copied over as well.

8.1.3.1. Modifying Properties of an Existing Material

Probably, the most common operation you will perform in the *Create/Edit Materials Dialog Box* (p. 1736) is the modification of properties for an existing material. The steps for this procedure are as follows:

1. Select the material you want to modify and the **Create/Edit...** button in the *Materials Task Page* (p. 1735).
2. In the *Create/Edit Materials Dialog Box* (p. 1736) select the type of material (**fluid**, **solid**, etc.) in the **Material Type** drop-down list.
3. Choose the material for which you want to modify properties, in the **Fluent Fluid Materials** drop-down list, **Fluent Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step.
4. Make the required changes to the properties listed in the **Properties** section of the dialog box. You can use the scroll bar to the right of the **Properties** section to scroll through the listed items.
5. Click on the **Change/Create** button to change the properties of the selected material to your new property settings.

To change the properties of an additional material, repeat the process described above. Click the **Change/Create** button after making changes to the properties for each material.

8.1.3.2. Renaming an Existing Material

Each material is identified by a name and a chemical formula (if one exists). You can change the name of a material, but not its chemical formula (unless you are creating a new material). The procedure for renaming a material is as follows:

1. Select the material you want to rename and the **Create/Edit...** button in the *Materials Task Page* (p. 1735).
2. In the *Create/Edit Materials Dialog Box* (p. 1736), choose the material for which you want to modify properties, in the **Fluent Fluid Materials** list, **Fluent Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step.
3. Enter the new name in the **Name** field at the top of the *Create/Edit Materials Dialog Box* (p. 1736).

   Important

   The maximum character length you can enter in the **Name** field is 29. If you enter a material name that is more than 29 characters long, ANSYS FLUENT will print an error message in the console window.

4. Click on the **Change/Create** button.

   A *Question Dialog Box* (p. 32) will appear, asking you if the original material should be overwritten.

   If you are renaming the original material, click **Yes** to overwrite it. If you were creating a new material, click **No** to retain the original material.

To rename another material, repeat the process described above. Click the **Change/Create** button after renaming each material.

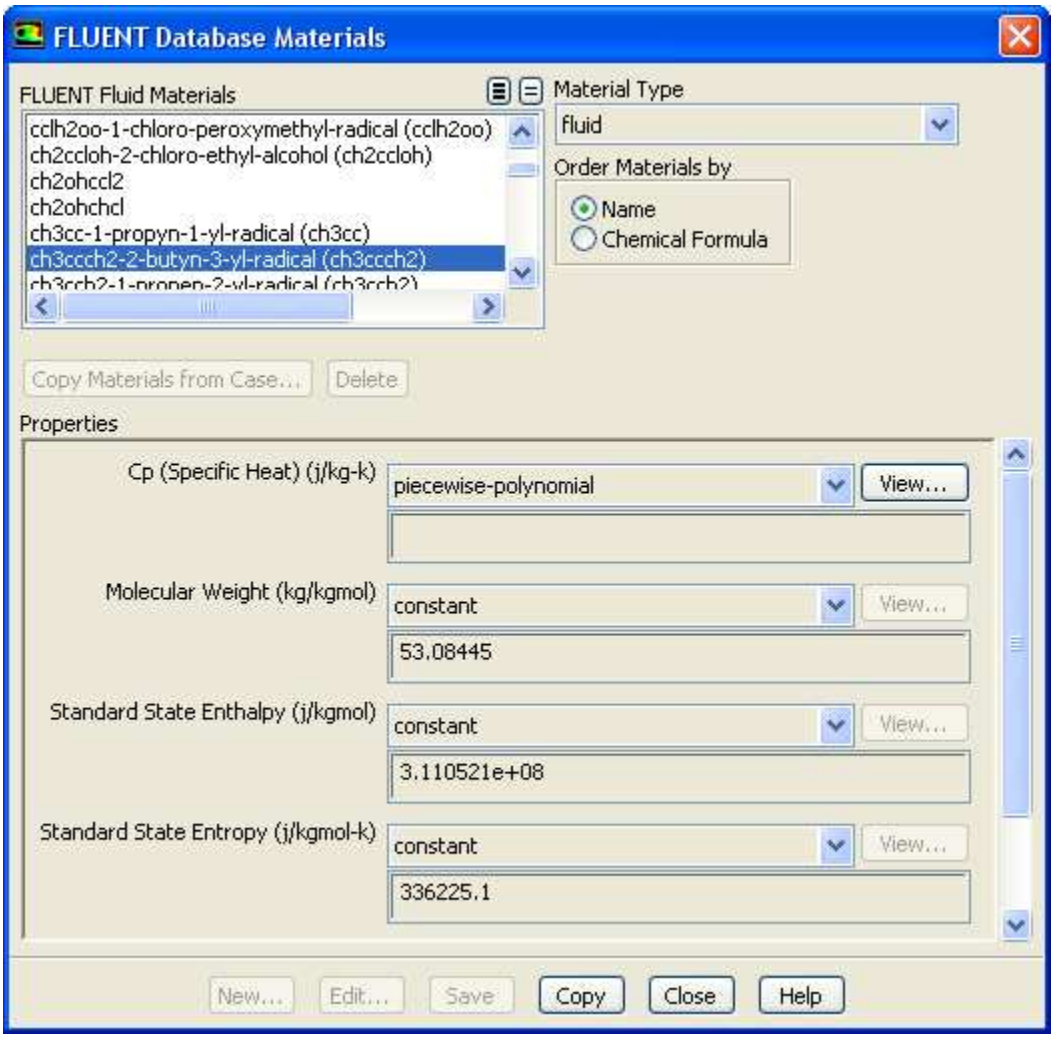
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.3.3. Copying Materials from the ANSYS FLUENT Database

The global (site-wide) materials database contains many commonly used fluid, solid, and mixture materials, with property data from several different sources [51] (p. 2211), [71] (p. 2212), [101] (p. 2214), [35] (p. 2210). To use one of these materials in your problem, copy it from the ANSYS FLUENT database to your local materials list. The procedure for copying a material is as follows:

1. Click on the **Fluent Database...** button in the *Create/Edit Materials Dialog Box* (p. 1736) to open the *FLUENT Database Materials Dialog Box* (p. 1744) (*Figure 8.2* (p. 389)).

Figure 8.2 Fluent Database Materials Dialog Box



2. Select the type of material (**fluid**, **solid**, etc.) in the **Material Type** drop-down list.
3. In the **Fluent Fluid Materials** list, **Fluent Solid Materials** list, or other similarly named list, choose the materials you wish to copy by clicking on them. The properties of the selected material will be displayed in the **Properties** area.
4. To check the material properties, use the scroll bar to the right of the **Properties** area to scroll through the listed items. For some properties, temperature-dependent functions are available in addition to the constant values. Select one of the function types in the drop-down list to the right of the property and the relevant parameters will be displayed. You cannot edit these values, but the dialog boxes in which they are displayed function in the same way as those used for setting temperature-dependent property functions (*Defining Properties Using Temperature-Dependent Functions* (p. 398)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*The inactive buttons in the FLUENT Database Materials Dialog Box (p. 1744) are operations that are applicable only for a user-defined database. These operations will be available when you click the **User-Defined Database...** button in the Create/Edit Materials Dialog Box (p. 1736).*

5. Click **Copy**. The materials and their properties will be downloaded from the database into your local list, and your copy of properties will now be displayed in the *Materials Task Page* (p. 1735).
6. Close the *FLUENT Database Materials Dialog Box* (p. 1744).

After copying a material from the database, you can modify its properties or change its name, as described earlier in this section. The original material in the database will not be affected by any changes made to your local copy of the material.

8.1.3.4. Creating a New Material

If the material you want to use is not available in the database, you can easily create a new material for the local list. This material will be available for use only for the current problem and will not be saved in the ANSYS FLUENT database. The procedure for creating a new material is as follows:

1. Select the **Create/Edit...** button in the *Materials Task Page* (p. 1735).
2. Select the new material type (**fluid**, **solid**, etc.) in the **Material Type** drop-down list. It does not matter which material is selected in the **Fluent Fluid Materials**, **Fluent Solid Materials**, or other similarly named list.
3. Enter the new material name in the **Name** field.

Important

The maximum character length you can enter in the **Name** field is 29. If you enter a material name that is more than 29 characters long, ANSYS FLUENT will print an error message in the console window.

4. Set the material's properties in the **Properties** area. If there are many properties listed, you may use the scroll bar to the right of the **Properties** area to scroll through the listed items.
5. Click on the **Change/Create** button. A *Question Dialog Box* (p. 32) will appear, asking you if the original material should be overwritten.
   a. Click on **No** to retain the original material and *add* your new material to the list. A dialog box will appear asking you to enter the chemical formula of your new material.
   b. Click **OK**, enter the formula if it is known. Else, leave the formula blank and click on **OK**. Select the **Change/Create** button and answer the **Question**.

   The *Materials Task Page* (p. 1735) will be updated to show the new material name and chemical formula in the **Fluid Materials** list (or **Fluent Solid Materials** or other similarly named list).

8.1.3.5. Saving Materials and Properties

All the materials and properties in your local list are saved in the case file when it is written. If you read this case file into a new solver session, all of your materials and properties will be available for use in the new session.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.3.6. Deleting a Material

If there are materials list that you no longer need, you can delete them:

1. Choose the material to be deleted in the *Materials Task Page* (p. 1735).
2. Click on the **Delete** button at the bottom of the *Materials Task Page* (p. 1735).

You can also delete materials in the *Create/Edit Materials Dialog Box* (p. 1736).

1. Select the material to be deleted and click the **Create/Edit...** button in the *Materials Task Page* (p. 1735).
2. In the *Create/Edit Materials Dialog Box* (p. 1736), select the type of material (**fluid**, **solid**, etc.) in the **Material Type** drop-down list.
3. Choose the material to be deleted in the **Fluent Fluid Materials** drop-down list, **Fluent Solid Materials** list, or other similarly named list. The list's name will be the same as the material type you selected in the previous step.
4. Click on the **Delete** button at the bottom of the *Materials Task Page* (p. 1735).

   Deleting materials contained in the global database.

8.1.3.7. Changing the Order of the Materials List

By default, the materials in your local list and those in the database are listed alphabetically by name (e.g., **air**, **atomic-oxygen (o)**, **carbon-dioxide (co2)**). If you prefer to list them alphabetically by chemical formula, select the **Chemical Formula** option under **Order Materials By**. The example materials listed, will now be in the order of: **air**, **co2 (carbon-dioxide)**, **o (atomic-oxygen)**. To change back to the alphabetical listing by name, choose the **Name** option under **Order Materials By**.

You may specify the ordering method separately for the *Create/Edit Materials Dialog Box* (p. 1736) and *FLUENT Database Materials Dialog Box* (p. 1744). For example, you can order the database materials list by name. Each dialog box has its own **Order Materials By** options.

8.1.4. Using a User-Defined Materials Database

In addition to the *FLUENT Database Materials Dialog Box* (p. 1744), you can also use or create a user-defined materials database using the *User-Defined Database Materials Dialog Box* (p. 1746). You can browse and do the following:

- select from existing user-defined databases
- copy materials from a user-defined database
- create a new database, create new materials
- add them to the user-defined database
- delete materials from the database
- copy materials from a case to a user-defined database
- view the database

The following sections will address each of these functionalities in detail.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.4.1. Opening a User-Defined Database

If you have a database of custom materials as `.scm` files with data saved in the specified format you can open these databases in ANSYS FLUENT and use them to define the materials in your problem setup.

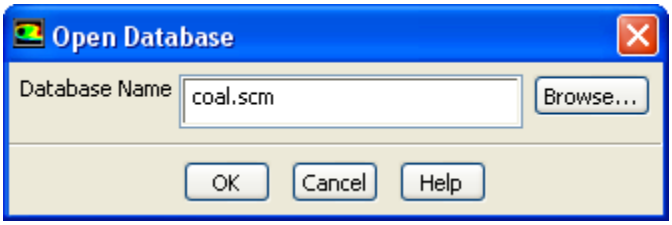
Examples:

The prescribed format for saving material properties information is shown here for air and aluminum. These files can be created in a text editor and saved with a `.scm` extension.

```
  ((air
  fluid
  (chemical-formula . #f)
  (density (constant . 1.225)
  (premixed-combustion 1.225 300))
  (specific-heat (constant . 1006.43))
  (thermal-conductivity (constant . 0.0242))
  (viscosity (constant . 1.7894e-05)
  (sutherland 1.7894e-05 273.11 110.56)
  (power-law 1.7894e-05 273.11 0.666))
  (molecular-weight (constant . 28.966))
  )
 (aluminum
  solid
  (chemical-formula . al)
  (density (constant . 2719))
  (specific-heat (constant . 871))
  (thermal-conductivity (constant . 202.4))
  (formation-entropy (constant . 164448.08))
  ))
```

To select a user-defined database, click the **User-Defined Database...** button in the *Create/Edit Materials Dialog Box* (p. 1736). This will open the *Open Database Dialog Box* (p. 1746).

Figure 8.3 Open Database Dialog Box



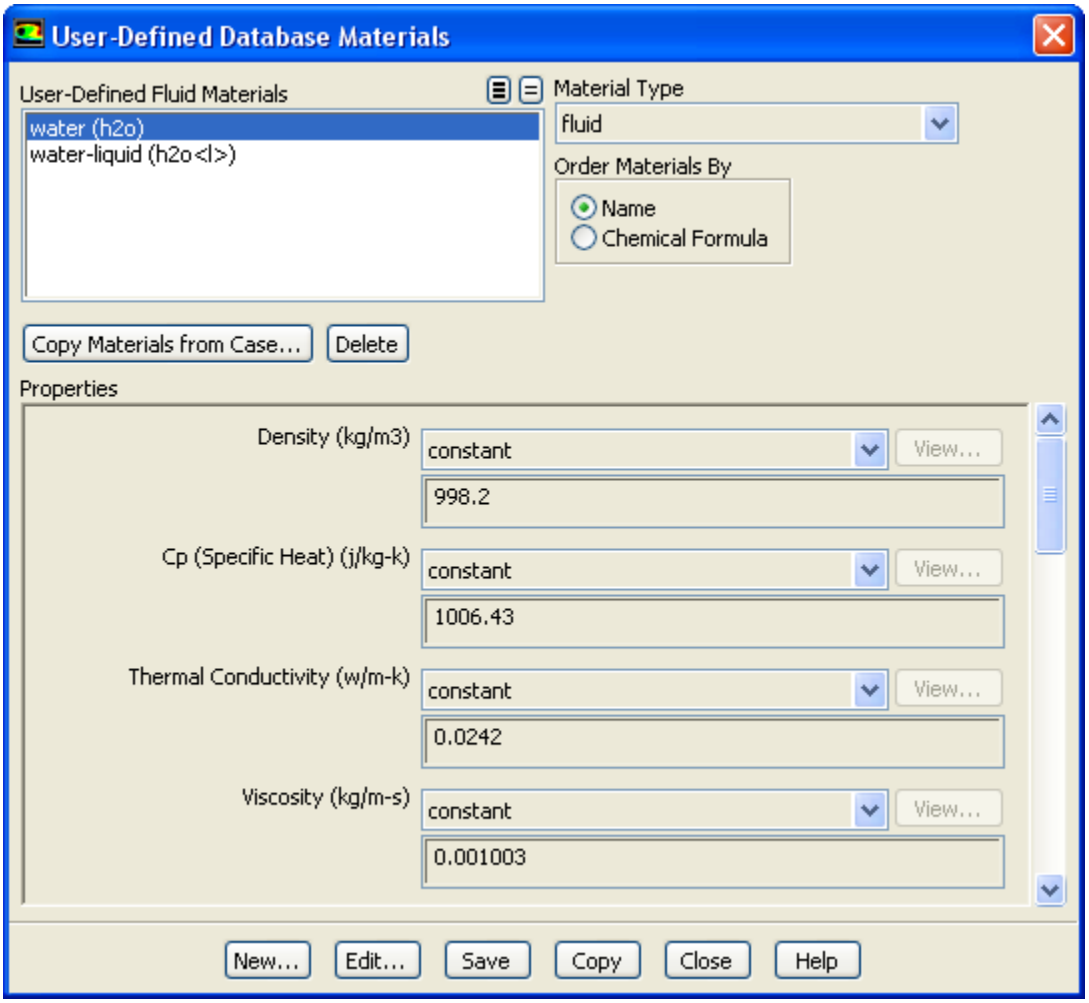
Click the **Browse...** button, select the database in *The **Select File** Dialog Box* (p. 33) that opens and click **OK**. Click **OK** in the *Open Database Dialog Box* (p. 1746) to open the *User-Defined Database Materials Dialog Box* (p. 1746).

8.1.4.2. Viewing Materials in a User-Defined Database

When an existing user-defined database is opened, the materials present in the database are listed in the *User-Defined Database Materials Dialog Box* (p. 1746). You can select the material type in the **Material Type** drop-down list and the corresponding materials will appear in the **User-Defined Fluid Materials**, **User-Defined Solid Materials** or other similarly named list. The name of the list will be the same as the material type you selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.4 User-Defined Database Materials Dialog Box



The properties of the selected material will appear in the **Properties** section of the dialog box. This dialog box is similar to the *FLUENT Database Materials Dialog Box* (p. 1744) in function and operation.

8.1.4.3. Copying Materials from a User-Defined Database

The procedure for copying a material from a custom database is as follows:

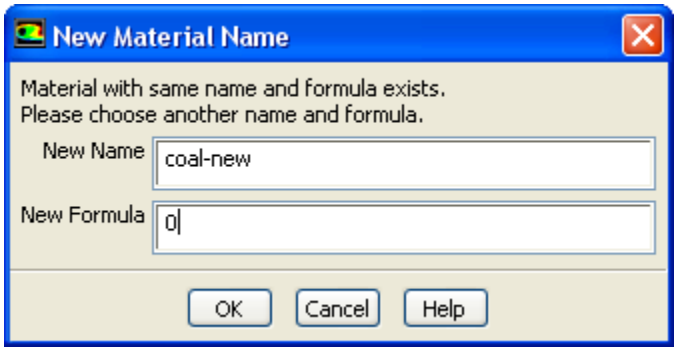
1. In the *Create/Edit Materials Dialog Box* (p. 1736), click the **User-Defined Database...** button and open the database from which you want to copy the material.
2. In the *User-Defined Database Materials Dialog Box* (p. 1746) of the selected database, select the type of material (**fluid**, **solid**, etc.) in the **Material Type** drop-down list.
3. In the **User-Defined Fluid Materials**list, **User-Defined Solid Materials** list, or other similarly named list (the list's name will be the same as the material type you selected in the previous step), choose the materials you wish to copy by clicking on them. The properties are displayed in the **Properties** area.
4. If you want to check the material properties, use the scroll bar to the right of the **Properties** area to scroll through the listed items.
5. Click on the **Copy** button. The selected materials and their properties will be copied from the database into your local list, and your copy of the properties will now be displayed in the *Materials Task Page* (p. 1735).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To copy all the materials from the database in one step, click the shaded icon to the right of the ***User-Defined Materials*** *title and click* ***Copy****.*

If a material with the same name is already defined in the case, ANSYS FLUENT will prompt you to enter a new name and formula in the *New Material Name Dialog Box* (p. 1750). Enter a new name and formula in the respective fields and click **OK** to make a local copy of the material.

Figure 8.5 New Material Name Dialog Box



6. Close the *User-Defined Database Materials Dialog Box* (p. 1746).

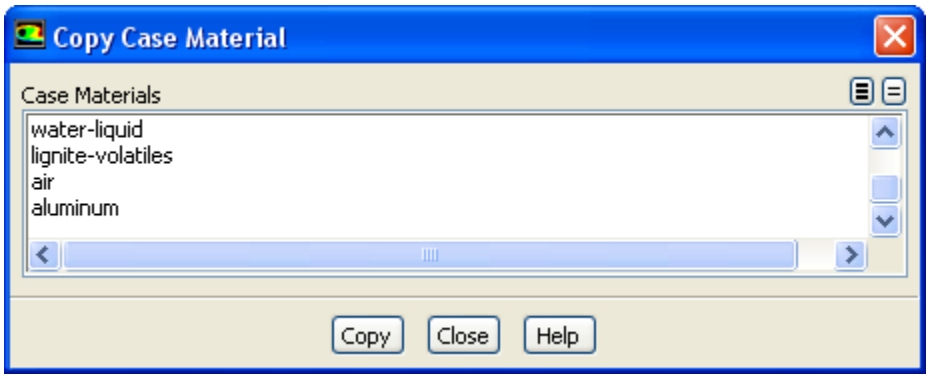
After copying a material from the database, you may modify its properties or change its name, as described earlier in *Using the Materials Task Page* (p. 387). The material in the database will not be affected by any changes you make to your local copy of the material.

8.1.4.4. Copying Materials from the Case to a User-Defined Database

You can copy materials from the case file to a database is as follows:

1. In the *Create/Edit Materials Dialog Box* (p. 1736), click **User-Defined Database...**.
2. In the *Open Database Dialog Box* (p. 1746), select the database to which you want to copy the material. If you want to create a new database, enter the name of the new database in the **Database Name** field and click **OK**. A *Question Dialog Box* (p. 32) will ask you to confirm if you want to create a new file. Click **Yes** to confirm.
3. In the *User-Defined Database Materials Dialog Box* (p. 1746), click **Copy Materials From Case....** This will open the *Copy Case Material Dialog Box* (p. 1748).

Figure 8.6 Copy Case Material Dialog Box



a. In the *Copy Case Material Dialog Box* (p. 1748), select the materials that you want to copy.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*To select all the materials, click on the shaded icon to the right of the **Case Materials** title. Clicking on the unshaded icon will deselect the selections in the list.*

b. Click **Copy** and close the dialog box.

Note

Do not copy materials one by one. This will result in previously copied materials getting overwritten by the new ones. Instead, select all the materials to be copied and click Copy.

8.1.4.5. Modifying Properties of an Existing Material

You can modify the properties of an existing material and use the modified material in the problem setup and save the modified material to the materials database.

1. In the *Materials Task Page* (p. 1735), click on the **User-Defined Database...** button and open the database that you want to use.
   a. In the *User-Defined Database Materials Dialog Box* (p. 1746) of the selected database, select the type of material (**fluid**, **solid**, etc.) in the **Material Type** drop-down list.
   b. In the **User-Defined Fluid Materials** list, **User-Defined Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step. Select the material to be modified.
   c. Click **Edit...** to open the *Material Properties Dialog Box* (p. 1748).
      i. In the **Materials Properties** list, select the property to be modified and click **Edit...** to open the *Edit Property Methods Dialog Box* (p. 1749).
      ii. Select the method to be modified in the **Material Properties** list of the *Edit Property Methods Dialog Box* (p. 1749) and click **Edit...** under **Edit Properties**, in order to modify the properties.
      iii. Make the changes in the corresponding method dialog box and click **OK**.
      iv. Click **Apply** in the *Material Properties Dialog Box* (p. 1748).
   d. To use the modified material in the problem setup, click **Copy** in the *User-Defined Database Materials Dialog Box* (p. 1746) and close the dialog box.
   e. To save the modified material to the database, click **Save**.

8.1.4.6. Creating a New Materials Database and Materials

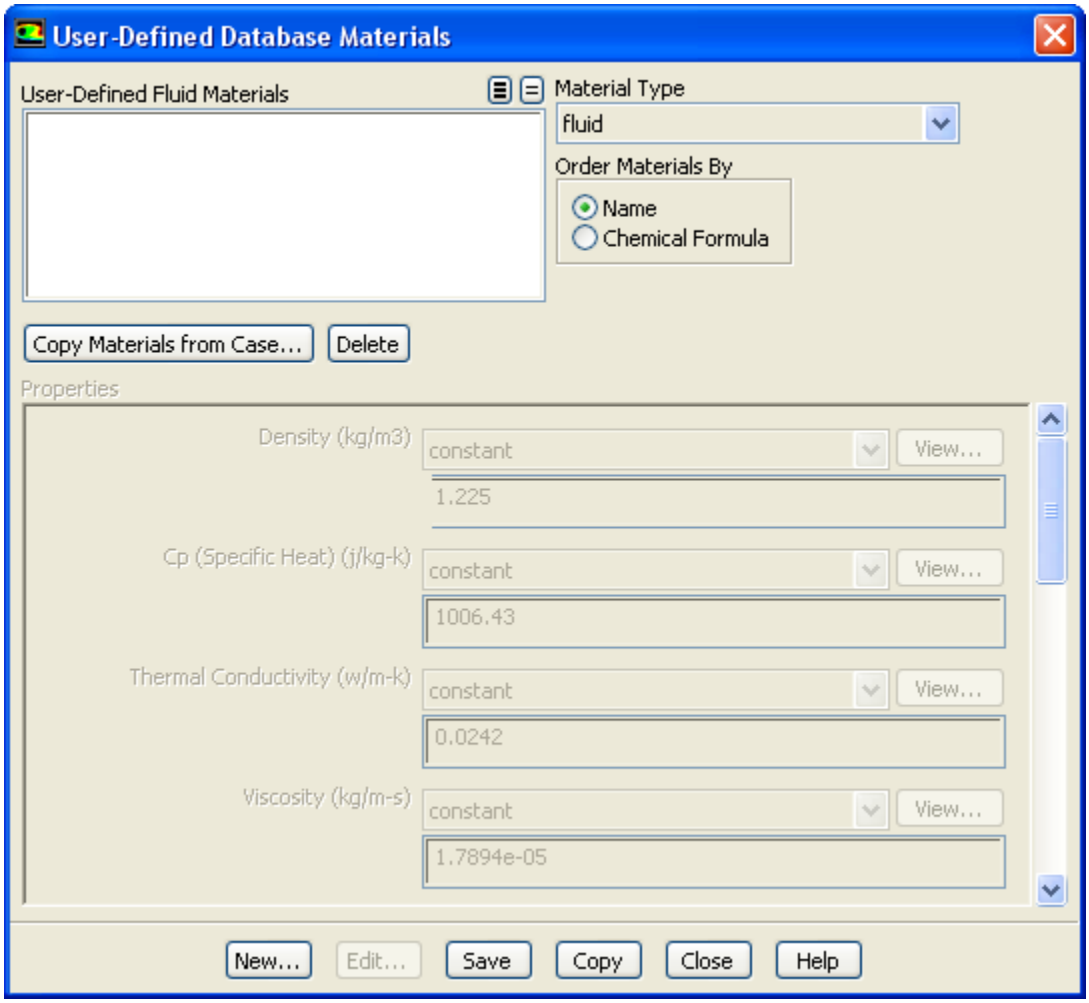
Using the *User-Defined Database Materials Dialog Box* (p. 1746), you can create a new materials to this database, and also create new materials to the database is as follows:

1. In the *Materials Task Page* (p. 1735), click **User-Defined Database...**.
2. In the *Open Database Dialog Box* (p. 1746), enter the name of the database that you are creating and click **OK**.
3. A dialog box will appear asking you confirm the creation of a new file. Click **Yes** to confirm.

   This will open a blank User-Defined Database Materials Dialog Box (p. 1746) (Figure 8.7 (p. 396)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

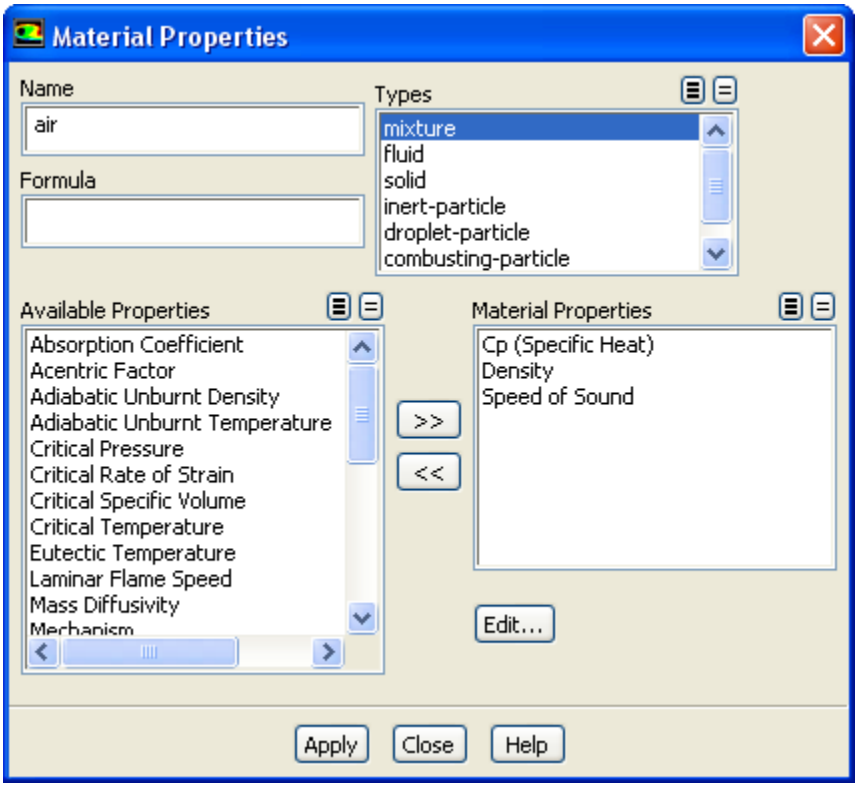
Figure 8.7 User-Defined Database Materials Dialog Box: Blank



4. Click **New...** in the *User-Defined Database Materials Dialog Box* (p. 1746). This will open a blank **Material Properties** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.8 Material Properties Dialog Box: Blank



a. In the *Material Properties Dialog Box* (p. 1748), under **Types**, select the material type. You can select from **fluid**, **solid**, **inert-particle**, **droplet-particle**, **combusting-particle**, and **mixture** materials.

b. Enter the name and formula (if required) of the material that you are creating in the **Name** and **Formula** fields.

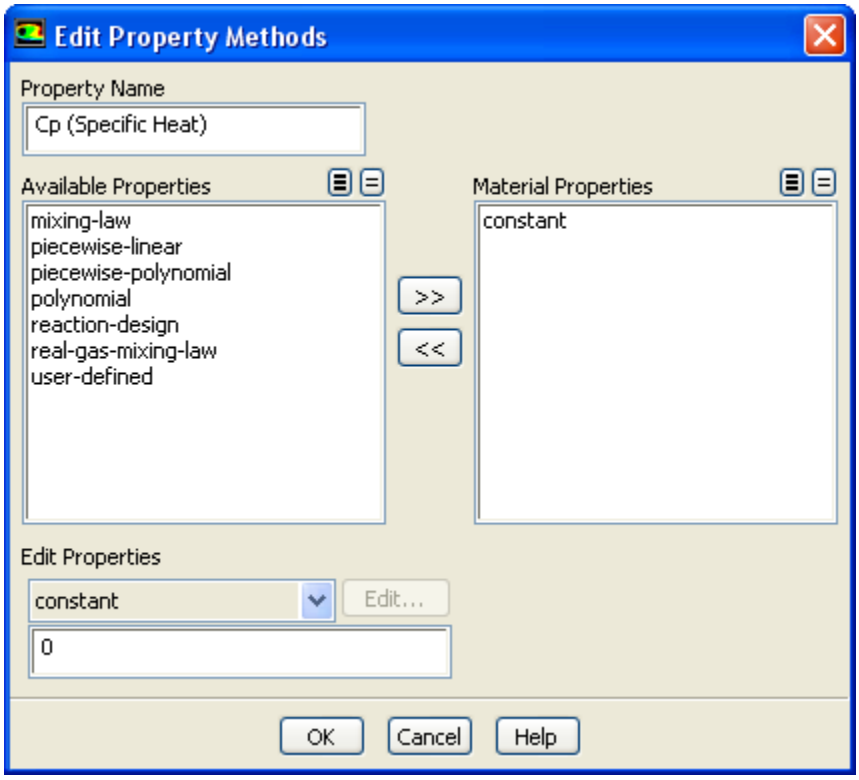
c. Depending on the type of material selected in the **Types** list, properties applicable to that material type will appear in the **Available Properties** list. Select the properties that are applicable for the material that you are defining by clicking on them.

d. Click the ≫ button to move these properties to the **Material Properties** list on the right and click **Apply**. You can use the ≪ button to move the property from the **Material Properties** list to the **Available Properties** list.

5. To edit the parameters that define a property, select the property in the **Material Properties** list and click **Edit...**. This opens the *Edit Property Methods Dialog Box* (p. 1749).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.9 Edit Property Methods Dialog Box



a. The methods that can be used to define the selected property are listed in the **Available Properties** list. You can select one or more methods and specify them for the material that you are defining, by selecting and moving them to the **Material Properties** list.

b. To modify each of these methods, you can select the method in the **Edit Properties** drop-down list and click **Edit...**. This will open the corresponding property dialog box, where you can modify the parameters used by the property method. Refer to *Defining Properties Using Temperature-Dependent Functions* (p. 398) to *Real Gas Models* (p. 451) for details of these properties, methods used to define the properties and the parameters for each method.

c. Click **OK** in the *Edit Property Methods Dialog Box* (p. 1749).

6. Click **Apply** in the *Material Properties Dialog Box* (p. 1748).

7. Click **Save** in the *User-Defined Database Materials Dialog Box* (p. 1746) to save the changes to the new materials database.

Similarly, you can also append new materials and click save to append these materials to the existing database.

8.1.4.7. Deleting Materials from a Database

To delete a material from a database, click the **User-Defined Database** button in the *Open Database Dialog Box* (p. 1746). Select the database and click **OK** in the *Open Database Dialog Box* (p. 1746). Select the **Material Type** and the materials that you want to delete in the **User-Defined Materials** list and click **Delete**. Click **Save** to save the database.

8.2. Defining Properties Using Temperature-Dependent Functions

Material properties can be defined as functions of temperature. For most properties, you can define a polynomial, piecewise-linear, or piecewise-polynomial function of temperature.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- polynomial:

$$\phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \quad \textbf{(8–1)}$$

- piecewise-linear:

$$\phi(T) = \phi_n + \frac{\phi_{n+1} - \phi_n}{T_{n+1} - T_n}(T - T_n) \quad \textbf{(8–2)}$$

where $1 \le n \le N$ and N is the number of segments

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < T_{max,1} : \phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2} : \phi(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \quad \textbf{(8–3)}$$

In the equations above, ϕ is the property.

Important

If you define a polynomial or piecewise-polynomial function of temperature, the temperature in the function is always in units of Kelvin or Rankine. If you use Celsius or Kelvin as the temperature unit, then polynomial coefficient values must be entered in terms of Kelvin. If you use Fahrenheit or Rankine as the temperature unit, enter the values in terms of Rankine.

Some properties have additional functions available and for some only a subset of these three functions can be used. See the section on the property in question to determine which temperature-dependent functions you can use.

For additional information, please see the following sections:

8.2.1. Inputs for Polynomial Functions
8.2.2. Inputs for Piecewise-Linear Functions
8.2.3. Inputs for Piecewise-Polynomial Functions
8.2.4. Checking and Modifying Existing Profiles

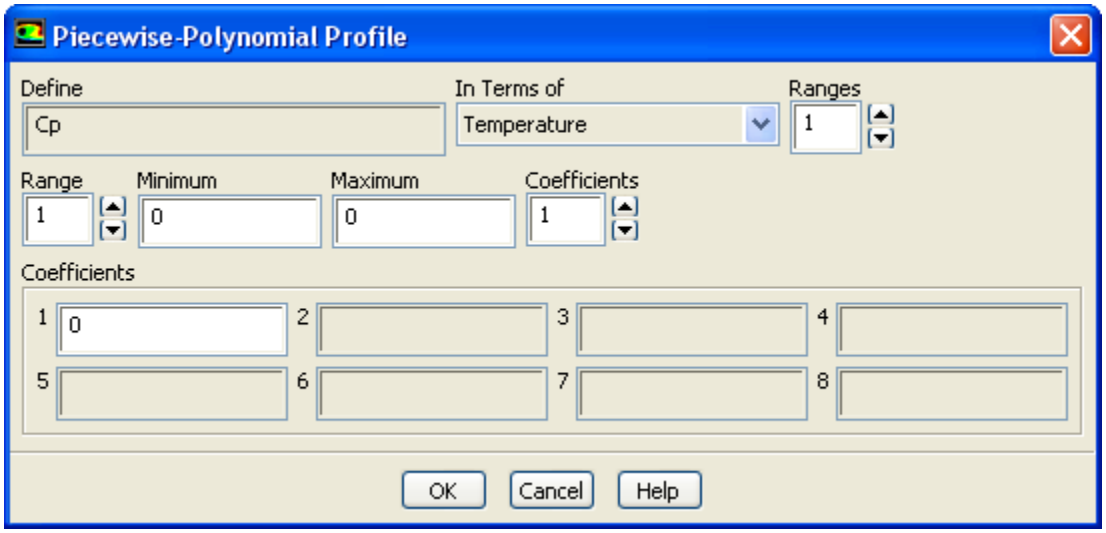
8.2.1. Inputs for Polynomial Functions

To define a polynomial function of temperature for a material property, do the following:

1. In the *Create/Edit Materials Dialog Box* (p. 1736), choose **polynomial** in the drop-down list to the right of the property name (e.g., **Viscosity**). The *Polynomial Profile Dialog Box* (p. 1751) (*Figure 8.10* (p. 400)) will open automatically.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.10 The Polynomial Profile Dialog Box



Since this is a modal dialog box, the solver will not allow you to do anything else until you perform the following steps.

a. Specify the number of **Coefficients** up to 8 coefficients are available. The number of coefficients defines the order of the polynomial. The default of 1 defines a polynomial of order 0. The property will be constant and equal to the single coefficient A_1. An input of 2 defines a polynomial of order 1 and the property will vary linearly with temperature and so on.

b. Define the coefficients. Coefficients **1**, **2**, **3**,... correspond to A_1, A_2, A_3,... in *Equation 8–1* (p. 399). The dialog box in *Figure 8.10* (p. 400)

$$\rho(T) = 1000 - 0.02T \tag{8–4}$$

Important

Note the restriction on the units for temperature, as described in the previous section.

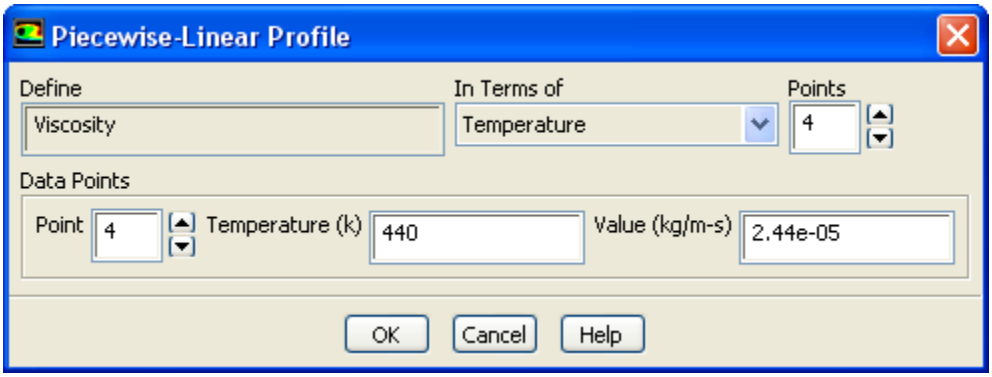
8.2.2. Inputs for Piecewise-Linear Functions

To define a piecewise-linear function of temperature for a material property, do the following:

1. In the *Create/Edit Materials Dialog Box* (p. 1736), choose **piecewise-linear** in the drop-down list to the right of the property name (e.g., **Viscosity**). The *Piecewise-Linear Profile Dialog Box* (p. 1751) (*Figure 8.11* (p. 401)) will open automatically.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.11 The Piecewise-Linear Profile Dialog Box



Since this is a modal dialog box, the solver will not allow you to do anything else until you perform the following steps.

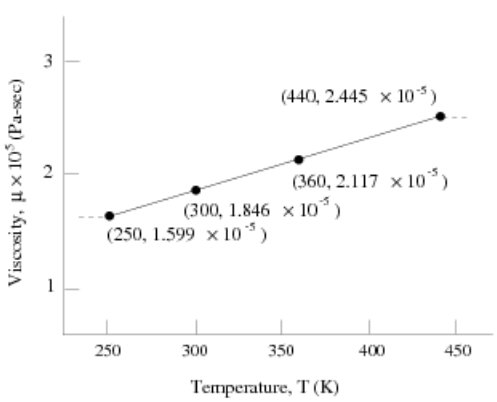
a. Set the number of **Points** defining the piecewise distribution.

b. Under **Data Points**, enter the data pairs for each point. First enter the independent and dependent variable values for **Point** 1, then increase the **Point** number and enter the appropriate values for each additional pair of variables. The pairs of points must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you. A maximum of 30 piecewise points can be defined for each property. The dialog box in *Figure 8.11* (p. 401) shows the final inputs for the profile depicted in *Figure 8.12* (p. 401).

Important

If the temperature exceeds the maximum **Temperature** (T_{max}) you have specified for the profile, ANSYS FLUENT will use the **Value** corresponding to T_{max}. If the temperature falls below the minimum **Temperature** (T_{min}) specified for your profile, ANSYS FLUENT will use the **Value** corresponding to T_{min}.

Figure 8.12 Piecewise-Linear Definition of Viscosity as a Function of Temperature



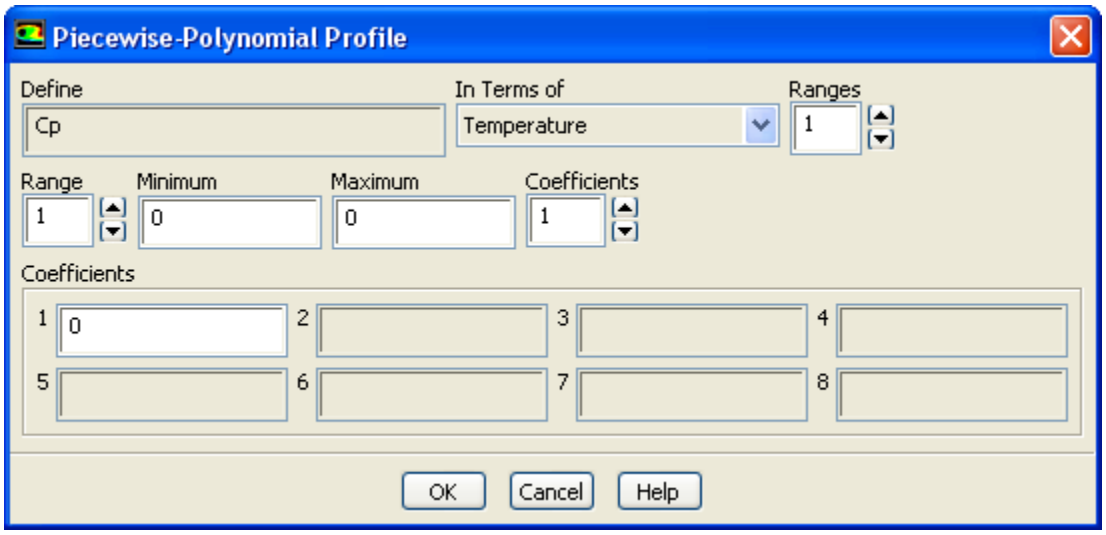
8.2.3. Inputs for Piecewise-Polynomial Functions

To define a piecewise-polynomial function of temperature for a material property, follow these steps:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the *Create/Edit Materials Dialog Box* (p. 1736), choose **piecewise-polynomial** in the drop-down list to the right of the property name (e.g., **Cp**). The *Piecewise-Polynomial Profile Dialog Box* (p. 1752) (*Figure 8.13* (p. 402)) will open automatically. Since this is a modal dialog box, first perform the following steps.

Figure 8.13 The Piecewise-Polynomial Profile Dialog Box



2. Specify the number of **Ranges**. For the example of *Equation 8–5* (p. 402), two ranges of temperatures are defined:

$$c_p(T) = \begin{cases} \text{for } 300 \le T < 1000: \\ 429.929 + 1.784T - 1.966\times10^{-3}T^2 + 1.297\times10^{-6}T^3 - 4.000\times10^{-10}T^4 \\ \text{for } 1000 \le T < 5000: \\ 841.377 + 0.593T - 2.415\times10^{-4}T^2 + 4.523\times10^{-8}T^3 - 3.153\times10^{-12}T^4 \end{cases} \quad \textbf{(8–5)}$$

You may define up to three ranges. The ranges must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you.

3. For the first range (**Range** = 1), specify the **Minimum** and **Maximum** temperatures, and the number of **Coefficients**. (Up to eight coefficients are available.) The number of coefficients defines the order of the polynomial. The default of 1 defines a polynomial of order 0. The property will be constant and equal to the single coefficient A_1. An input of 2 defines a polynomial of order 1. The property will vary linearly with temperature and so on.

4. Define the coefficients. Coefficients **1**, **2**, **3**,... correspond to A_1, A_2, A_3,... in *Equation 8–3* (p. 399). The dialog box in *Figure 8.13* (p. 402) shows the inputs for the first range of *Equation 8–5* (p. 402).

5. Increase the value of **Range** and enter the **Minimum** and **Maximum** temperatures, number of **Coefficients**, and the **Coefficients** (B_1, B_2, B_3,...) for the next range. Repeat if there is a third range.

8.2.4. Checking and Modifying Existing Profiles

If you want to check or change the coefficients, data pairs, or ranges for a previously-defined profile, click on the **Edit...** button to the right of the property name. The appropriate dialog box will open, and you can check or modify the inputs as desired.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

In the *FLUENT Database Materials Dialog Box* (p. 1744), you cannot edit the profiles, but you can examine them by clicking on the **View...** button (instead of the **Edit...** button.)

8.3. Density

ANSYS FLUENT provides several options for definition of the fluid density:

- constant density
- temperature and/or composition dependent density

Each of these input options and the governing physical models are explained in the following sections. In all cases, you will define the **Density** in the *Create/Edit Materials Dialog Box* (p. 1736).

Materials

Density can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736)). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

For additional information, please see the following sections:

8.3.1. Defining Density for Various Flow Regimes
8.3.2. Input of Constant Density
8.3.3. Inputs for the Boussinesq Approximation
8.3.4. Density as a Profile Function of Temperature
8.3.5. Incompressible Ideal Gas Law
8.3.6. Ideal Gas Law for Compressible Flows
8.3.7. Composition-Dependent Density for Multicomponent Mixtures

8.3.1. Defining Density for Various Flow Regimes

The selection of density in ANSYS FLUENT is very important. Set the density relationship based on your flow regime.

- For compressible flows, the ideal gas law is the appropriate density relationship.
- For incompressible flows, you may choose one of the following methods:
  - Constant density, if you do not want density to be a function of temperature.
  - The incompressible ideal gas law, when pressure variations are small enough that the flow is fully incompressible but you wish to use the ideal gas law to express the relationship between density and temperature (e.g., for a natural convection problem).
  - Density as a polynomial, piecewise-linear, or piecewise-polynomial function of temperature, when the density is a function of temperature only, as in a natural convection problem.
  - The Boussinesq model, for natural convection problems involving small changes in temperature.

8.3.1.1. Mixing Density Relationships in Multiple-Zone Models

If your model has multiple fluid zones that use different materials, you should be aware of the following:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For calculations with the pressure-based solver that do not use one of the general multiphase models, the compressible ideal gas law cannot be mixed with any other density methods. This means that if the compressible ideal gas law is used for one material, it must be used for all materials.

  This restriction does not apply to the density-based solvers.
- There is only one specified operating pressure and one specified operating temperature. This means that if you are using the ideal gas law for more than one material, they will share the same operating pressure. If you are using the Boussinesq model for more than one material, they will share the same operating temperature.

8.3.2. Input of Constant Density

If you want to define the density of the fluid as a constant, select **constant** in the **Density** drop-down list under **Properties** in the *Create/Edit Materials Dialog Box* (p. 1736). Enter the value of density for the material.

For the default fluid (air), the density is 1.225 kg/m^3.

8.3.3. Inputs for the Boussinesq Approximation

To enable the Boussinesq approximation for density, choose **boussinesq** from the **Density** drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736) and specify a constant value for **Density**. You will also need to set the **Thermal Expansion Coefficient**, as well as relevant operating conditions, as described in *The Boussinesq Model* (p. 697).

8.3.4. Density as a Profile Function of Temperature

If you are modeling a problem that involves heat transfer, you can define the density as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$\rho(T) = \rho_n + \frac{\rho_{n+1} - \rho_n}{T_{n+1} - T_n}(T - T_n) \tag{8–6}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < \mathrm{T}_{max,1} : \rho(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < \mathrm{T}_{max,2} : \rho(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{8–7}$$

- polynomial:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\rho(T) = A_1 + A_2 T + A_3 T^2 + \ldots \quad \textbf{(8–8)}$$

For one of the these methods, select **piecewise-linear**, **piecewise-polynomial**, or **polynomial** in the **Density** drop-down list. You can enter the data pairs (T_n, ρ_n), ranges and coefficients, or coefficients that describe these functions using the *Create/Edit Materials Dialog Box* (p. 1736), as described in *Defining Properties Using Temperature-Dependent Functions* (p. 398).

8.3.5. Incompressible Ideal Gas Law

In ANSYS FLUENT, if you choose to define the density using the ideal gas law for an incompressible flow, the solver will compute the density as

$$\rho = \frac{p_{op}}{\frac{R}{M_w} T} \quad \textbf{(8–9)}$$

where,

R = the universal gas constant

M_w = the molecular weight of the gas

p_{op} = the operating pressure

In this form, the density depends only on the operating pressure and not on the local relative pressure field.

8.3.5.1. Density Inputs for the Incompressible Ideal Gas Law

The inputs for the incompressible ideal gas law are as follows:

1. Enable the ideal gas law for an incompressible fluid by choosing **incompressible-ideal-gas** from the drop-down list to the right of **Density** in the *Create/Edit Materials Dialog Box* (p. 1736).

   Specify the incompressible ideal gas law individually for each material that you want to use it for. See *Composition-Dependent Density for Multicomponent Mixtures* (p. 407) for information on specifying the incompressible ideal gas law for mixtures.

2. Set the operating pressure by defining the **Operating Pressure** in the *Operating Conditions Dialog Box* (p. 1806).

   Cell Zone Conditions → **Operating Conditions...**

   Important

   By default, operating pressure is set to 101325 Pa. The input of the operating pressure is of great importance when you are computing density with the ideal gas law. See *Operating Pressure* (p. 449) for recommendations on setting appropriate values for the operating pressure.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set the molecular weight of the homogeneous or single-component fluid (if no chemical species transport equations are to be solved), or the molecular weights of each fluid material (species) in a multicomponent mixture. For each fluid material, enter the value of the **Molecular Weight** in the *Create/Edit Materials Dialog Box* (p. 1736).

8.3.6. Ideal Gas Law for Compressible Flows

For compressible flows, the gas law is as following:

$$\rho = \frac{p_{op} + p}{\frac{R}{M_w} T} \tag{8–10}$$

where,

p = the local relative (or gauge) pressure predicted by ANSYS FLUENT

p_{op} = the operating pressure

8.3.6.1. Density Inputs for the Ideal Gas Law for Compressible Flows

The inputs for the ideal gas law are as follows:

1. Enable the ideal gas law for a compressible fluid by choosing **ideal-gas** from the drop-down list to the right of **Density** in the *Create/Edit Materials Dialog Box* (p. 1736).

   Specify the ideal gas law individually for each material that you want to use it for. See *Composition-Dependent Density for Multicomponent Mixtures* (p. 407) for information on specifying the ideal gas law for mixtures.

2. Set the operating pressure by defining the **Operating Pressure** in the *Operating Conditions Dialog Box* (p. 1806).

   Cell Zone Conditions → **Operating Conditions...**

   Important

   The input of the operating pressure is of great importance when you are computing density with the ideal gas law. *Equation 8–10* (p. 406) notes that the operating pressure is added to the relative pressure field computed by the solver, yielding the absolute static pressure. See *Operating Pressure* (p. 449) for recommendations on setting appropriate values for the operating pressure. By default, operating pressure is set to 101325 Pa.

3. Set the molecular weight of the homogeneous or single-component fluid (if no chemical species transport equations are to be solved), or the molecular weights of each fluid material (species) in a multicomponent mixture. For each fluid material, enter the value of the **Molecular Weight** in the *Create/Edit Materials Dialog Box* (p. 1736).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.7. Composition-Dependent Density for Multicomponent Mixtures

If you are solving species transport equations, set properties for the mixture material and for the constituent fluids (species), as described in detail in *Defining Properties for the Mixture and Its Constituent Species* (p. 805). To define a composition-dependent density for a mixture, do the following:

1. Select the density method:
   - For non-ideal-gas mixtures, select the **volume-weighted-mixing-law** method for the mixture material in the drop-down list to the right of **Density** in the *Create/Edit Materials Dialog Box* (p. 1736).
   - If you are modeling compressible flow, select **ideal-gas** for the mixture material in the drop-down list to the right of **Density** in the *Create/Edit Materials Dialog Box* (p. 1736).
   - If you are modeling incompressible flow using the ideal gas law, select **incompressible-ideal-gas** for the mixture material in the **Density** drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736).
   - If you have a user-defined function that you want to use to model the density, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. (Note that only the constant, the polynomial methods and the user-defined methods are available.)
2. Click **Change/Create**.
3. If you have selected **volume-weighted-mixing-law**, define the density for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent densities for the individual species.
4. If you selected **user-defined-mixing-law**, define the density for each of the fluid materials that comprise the mixture. You may define constant, or (if applicable) temperature-dependent densities, or user-defined densities for the individual species. More information on defining properties with user-defined functions can be found in the UDF Manual.

If you are modeling a non-ideal-gas mixture, ANSYS FLUENT will compute the mixture density as

$$\rho = \frac{1}{\sum_i \frac{Y_i}{\rho_i}} \tag{8–11}$$

where Y_i is the mass fraction and ρ_i is the density of species i.

For compressible flows, the gas law has the following form:

$$\rho = \frac{p_{op} + p}{RT \sum_i \frac{Y_i}{M_{w,i}}} \tag{8–12}$$

where,

p = the local relative (or gauge) pressure predicted by ANSYS FLUENT

R = the universal gas constant

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

In ANSYS FLUENT, if you choose to define the density using the ideal gas law for an incompressible flow, the solver will compute the density as

$$\rho = \frac{p_{op}}{RT\sum_i \frac{Y_i}{M_{w,i}}} \tag{8–13}$$

where,

R = the universal gas constant

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

8.4. Viscosity

ANSYS FLUENT provides several options for definition of the fluid viscosity:

- constant viscosity
- temperature dependent and/or composition dependent viscosity
- kinetic theory
- non-Newtonian viscosity
- user-defined function

Each of these input options and the governing physical models are detailed in this section. (User-defined functions are described in the UDF Manual). In all cases, define the **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736).

Materials

Viscosities are input as dynamic viscosity (μ) in units of kg/m-s in SI units or lb_m/ft-s in British units. ANSYS FLUENT does not ask for input of the kinematic viscosity (ν).

Viscosity can also be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736).). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

For additional information, please see the following sections:

8.4.1. Input of Constant Viscosity
8.4.2. Viscosity as a Function of Temperature
8.4.3. Defining the Viscosity Using Kinetic Theory

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures
8.4.5. Viscosity for Non-Newtonian Fluids

8.4.1. Input of Constant Viscosity

If you want to define the viscosity of your fluid as a constant, select **constant** in the **Viscosity** drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736), and enter the value of viscosity for the fluid.

For the default fluid (air), the viscosity is 1.7894×10^{-5} kg/m-s.

8.4.2. Viscosity as a Function of Temperature

If you are modeling a problem that involves heat transfer, you can define the viscosity as a function of temperature. Five types of functions are available.

- piecewise-linear:

$$\mu(T) = \mu_n + \frac{\mu_{n+1} - \mu_n}{T_{n+1} - T_n}(T - T_n) \tag{8–14}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < \mathrm{T}_{max,1} : \mu(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < \mathrm{T}_{max,2} : \mu(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{8–15}$$

- polynomial:

$$\mu(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{8–16}$$

- Sutherland's law
- power law

Important

The power law described here is different from the non-Newtonian power law described in *Viscosity for Non-Newtonian Fluids* (p. 413).

For one of the first three, select **piecewise-linear**, **piecewise-polynomial**, **polynomial** in the **Viscosity** drop-down list and then enter the data pairs (T_n, μ_n), ranges and coefficients, or coefficients that describe these functions *Defining Properties Using Temperature-Dependent Functions* (p. 398). For Sutherland's law or the power law, choose **sutherland** or **power-law** respectively in the drop-down list and enter the parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2.1. Sutherland Viscosity Law

Sutherland's viscosity law resulted from a kinetic theory by Sutherland (1893) using an idealized intermolecular-force potential. The formula is specified using two or three coefficients.

Sutherland's law with two coefficients has the form

$$\mu = \frac{C_1 T^{3/2}}{T + C_2} \tag{8–17}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

C_1 and C_2 = the coefficients

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

For air at moderate temperatures and pressures, $C_1 = 1.458 \times 10^{-6}$ kg/m-s-$K^{1/2}$, and $C_2 = 110.4$ K.

Sutherland's law with three coefficients has the form

$$\mu = \mu_0 \left(\frac{T}{T_0} \right)^{3/2} \frac{T_0 + S}{T + S} \tag{8–18}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

μ_0 = reference value in kg/m-s

T_0 = reference temperature in K

S = an effective temperature in K (Sutherland constant)

For air at moderate temperatures and pressures, $\mu_0 = 1.716 \times 10^{-5}$ kg/m-s, T_0 = 273.11 K, and S = 110.56 K.

8.4.2.1.1. Inputs for Sutherland's Law

To use Sutherland's law, choose **sutherland** in the drop-down list to the right of **Viscosity**. The *Sutherland Law Dialog Box* (p. 1754) will open, and you can enter the coefficients as follows:

1. Select the **Two Coefficient Method**or the **Three Coefficient Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Use SI units if you choose the two-coefficient method.

2. For the **Two Coefficient Method**, set **C1** and **C2**. For the **Three Coefficient Method**, set the **Reference Viscosity** μ_0, the **Reference Temperature** T_0, and the **Effective Temperature** S.

8.4.2.2. Power-Law Viscosity Law

Another common approximation for the viscosity of dilute gases is the power-law form. For dilute gases at moderate temperatures, this form is considered to be slightly less accurate than Sutherland's law.

A power-law viscosity law with two coefficients has the form

$$\mu = BT^n \tag{8–19}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

B = a dimensional coefficient

For air at moderate temperatures and pressures, $B = 4.093 \times 10^{-7}$, and $n = 2/3$.

A power-law viscosity law with three coefficients has the form

$$\mu = \mu_0 \left(\frac{T}{T_0} \right)^n \tag{8–20}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

T_0 = a reference value in K

μ_0 = a reference value in kg/m-s

For air at moderate temperatures and pressures, $\mu_0 = 1.716 \times 10^{-5}$ kg/m-s, $T_0 = 273$ K, and $n = 2/3$.

Important

The non-Newtonian power law for viscosity is described in *Viscosity for Non-Newtonian Fluids* (p. 413).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2.2.1. Inputs for the Power Law

To use the power law, choose **power-law** in the drop-down list to the right of **Viscosity**. The *Power Law Dialog Box* (p. 1755) will open, and you can enter the coefficients as follows:

1. Select the **Two Coefficient Method**or the **Three Coefficient Method**.

   Important

   Note that you must use SI units if you choose the two-coefficient method.

2. For the **Two Coefficient Method**, set **B** and the **Temperature Exponent**n. For the **Three Coefficient Method**, set the **Reference Viscosity**μ_0, the **Reference Temperature** T_0, and the **Temperature Exponent**n.

8.4.3. Defining the Viscosity Using Kinetic Theory

If you are using the gas law (as described in *Density* (p. 403)), you have the option to define the fluid viscosity using kinetic theory as

$$\mu = 2.67 \times 10^{-6} \frac{\sqrt{M_w T}}{\sigma^2 \Omega_\mu} \tag{8–21}$$

where μ is in units of kg/m-s, T is in units of Kelvin, σ is in units of Angstroms, and $\Omega_\mu = \Omega_\mu(T^*)$ where

$$T^* = \frac{T}{(\varepsilon / k_B)} \tag{8–22}$$

The Lennard-Jones parameters, σ and ε / k_B, are inputs to the kinetic theory calculation that you supply by selecting **kinetic-theory** from the drop-down list to the right of **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). The solver will use these kinetic theory inputs in *Equation 8–21* (p. 412) to compute the fluid viscosity. See *Kinetic Theory Parameters* (p. 448) for details about these inputs.

8.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent viscosity. (Note that you can also define the viscosity of the mixture as a constant value or a function of temperature.)

To define a composition-dependent viscosity for a mixture, follow these steps:

1. For the mixture material, choose **mass-weighted-mixing-law** or, if you are using the ideal gas law for density, **ideal-gas-mixing-law** in the drop-down list to the right of **Viscosity**. If you have a user-defined function that you want to use to model the viscosity, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.
2. Click **Change/Create**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the viscosity for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent viscosities for the individual species. You may also use kinetic theory for the individual viscosities, or specify a non-Newtonian viscosity, if applicable.
4. If you selected **user-defined-mixing-law**, define the viscosity for each of the fluid materials that comprise the mixture. You may define constant, or (if applicable) temperature-dependent viscosities, or user-defined viscosities for the individual species. More information on defining properties with user-defined functions can be found in the UDF Manual.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. (Note that only the constant, the polynomial methods and the user-defined methods are available.)

If you are using the ideal gas law, the solver will compute the mixture viscosity based on kinetic theory as

$$\mu = \sum_i \frac{X_i \mu_i}{\sum_j X_j \phi_{ij}} \tag{8–23}$$

where

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{1/2} \left(\frac{M_{w,j}}{M_{w,i}}\right)^{1/4}\right]^2}{\left[8\left(1 + \frac{M_{w,i}}{M_{w,j}}\right)\right]^{1/2}} \tag{8–24}$$

and X_i is the mole fraction of species i.

For non-ideal gas mixtures, the mixture viscosity is computed based on a simple mass fraction average of the pure species viscosities:

$$\mu = \sum_i Y_i \mu_i \tag{8–25}$$

8.4.5. Viscosity for Non-Newtonian Fluids

For incompressible Newtonian fluids, the shear stress is proportional to the rate-of-deformation tensor $\overline{\overline{D}}$:

$$\overline{\overline{\tau}} = \mu \overline{\overline{D}} \tag{8–26}$$

where $\overline{\overline{D}}$ is defined by

$$\overline{\overline{D}} = \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) \tag{8–27}$$

and μ is the viscosity, which is independent of $\overline{\overline{D}}$.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For some non-Newtonian fluids, the shear stress can similarly be written in terms of a non-Newtonian viscosity η:

$$\overline{\overline{\tau}} = \eta(\overline{\overline{D}})\overline{\overline{D}} \tag{8–28}$$

In general, η is a function of all three invariants of the rate-of-deformation tensor $\overline{\overline{D}}$. However, in the non-Newtonian models available in ANSYS FLUENT, η is considered to be a function of the shear rate $\dot{\gamma}$ only. $\dot{\gamma}$ is related to the second invariant of $\overline{\overline{D}}$ and is defined as

$$\dot{\gamma} = \sqrt{\frac{1}{2}\overline{\overline{D}} : \overline{\overline{D}}} \tag{8–29}$$

8.4.5.1. Temperature Dependent Viscosity

If the flow is non-isothermal, then the temperature dependence on the viscosity can be included along with the shear rate dependence. In this case, the total viscosity consists of two parts and is calculated as

$$\mu = \eta(\dot{\gamma})H(T) \tag{8–30}$$

where H(T) is the temperature dependence, known as the Arrhenius law.

$$H(T) = exp\left[\alpha\left(\frac{1}{T-T_0} - \frac{1}{T_\alpha - T_0}\right)\right] \tag{8–31}$$

where α is the ratio of the activation energy to the thermodynamic constant and T_α is a reference temperature for which H(T) = 1. T_0, which is the temperature shift, is set to 0 by default, and corresponds to the lowest temperature that is thermodynamically acceptable. Therefore T_α and T_0 are absolute temperatures. Temperature dependence is only included when the energy equation is enabled. Set the parameter α to 0 when you want temperature dependence to be ignored, even when the energy equation is solved.

ANSYS FLUENT provides four options for modeling non-Newtonian flows:

- power law
- Carreau model for pseudo-plastics
- Cross model
- Herschel-Bulkley model for Bingham plastics

Important

- Note that the models listed above are not available when modeling turbulent flow.
- Note that the non-Newtonian power law described below is different from the power law described in *Power-Law Viscosity Law* (p. 411).
- Non-Newtonian model for single phase is available for the mixture model and it is recommended that this should be attached to the primary phase.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Appropriate values for the input parameters for these models can be found in the literature (e.g., [92] (p. 2213)).

8.4.5.2. Power Law for Non-Newtonian Viscosity

If you choose **non-newtonian-power-law** in the drop-down list to the right of **Viscosity**, non-Newtonian flow will be modeled according to the following power law for the non-Newtonian viscosity:

$$\eta = k\dot{\gamma}^{n-1}H(T) \tag{8–32}$$

where k and n are input parameters. k is a measure of the average viscosity of the fluid (the consistency index); n is a measure of the deviation of the fluid from Newtonian (the power-law index). The value of n determines the class of the fluid:

$n = 1 \rightarrow$ Newtonian fluid

$n > 1 \rightarrow$ shear-thickening (dilatant fluids)

$n < 1 \rightarrow$ shear-thinning (pseudo-plastics)

8.4.5.2.1. Inputs for the Non-Newtonian Power Law

To use the non-Newtonian power law, choose **non-newtonian-power-law** in the drop-down list to the right of **Viscosity**. The *Non-Newtonian Power Law Dialog Box* (p. 1755) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Consistency Index**k, **Power-Law Index**n, **Minimum** and **Maximum Viscosity Limit**, **Reference Temperature** T_{alpha}, and **Activation Energy/R**, which is the ratio of the activation energy to the thermodynamic constant α.

8.4.5.3. The Carreau Model for Pseudo-Plastics

The power law model described in *Equation 8–32* (p. 415) results in a fluid viscosity that varies with shear rate. For $\dot{\gamma} \rightarrow 0, \eta \rightarrow \eta_0$, and for $\dot{\gamma} \rightarrow \infty, \eta \rightarrow \eta_\infty$, where η_0 and η_∞ are, respectively, the upper and lower limiting values of the fluid viscosity.

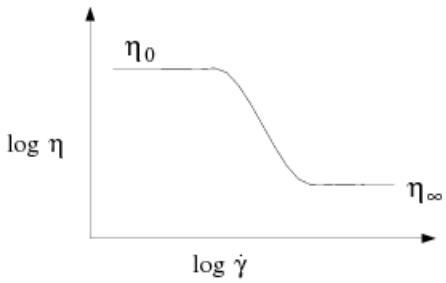
The Carreau model attempts to describe a wide range of fluids by the establishment of a curve-fit to piece together functions for both Newtonian and shear-thinning ($n < 1$) non-Newtonian laws. In the Carreau model, the viscosity is

$$\eta = H(T)\left(\eta_\infty + \left(\eta_0 - \eta_\infty\right)\left[1 + \gamma^2\lambda^2\right]^{(n-1)/2}\right) \tag{8–33}$$

and the parameters $n, \lambda, T_\alpha, \eta_0$, and η_∞ are dependent upon the fluid. λ is the time constant, n is the power-law index (as described above for the non-Newtonian power law), η_0 and η_∞ are, respectively, the zero- and infinite-shear viscosities, T_α is the reference temperature, and α is the ratio of the activation energy to thermodynamic constant. *Figure 8.14* (p. 416) shows how viscosity is limited by η_0 and η_∞ at low and high shear rates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

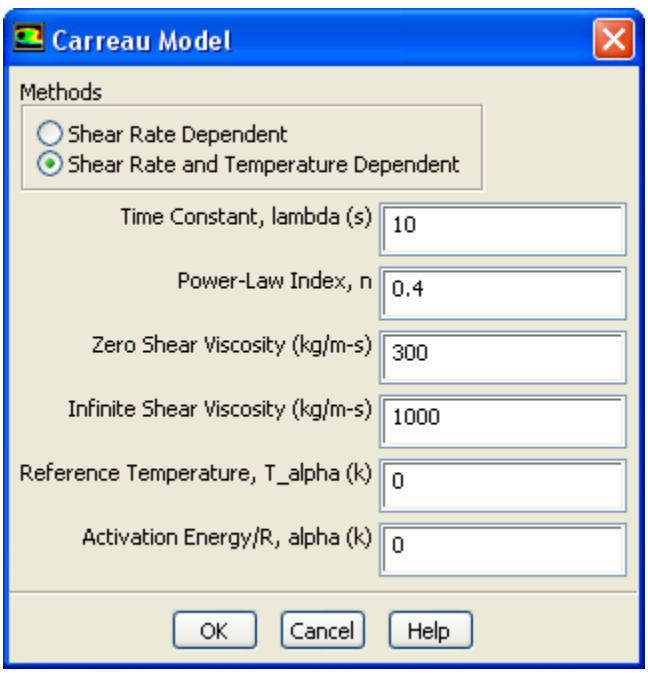
Figure 8.14 Variation of Viscosity with Shear Rate According to the Carreau Model



8.4.5.3.1. Inputs for the Carreau Model

To use the Carreau model, choose **carreau** in the drop-down list to the right of **Viscosity**. The *Carreau Model Dialog Box* (p. 1756) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Time Constant** λ, **Power-Law Index** n, **Reference Temperature** T_α, **Zero Shear Viscosity** η_0, **Infinite Shear Viscosity** η_∞, and **Activation Energy/R** α.

Figure 8.15 The Carreau Model Dialog Box



8.4.5.4. Cross Model

The Cross model for viscosity is:

$$\eta = H(T)\frac{\eta_0}{1+(\lambda\dot{\gamma})^{1-n}} \qquad \textbf{(8–34)}$$

where,

η_0 = zero-shear-rate viscosity

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

λ = natural time (i.e.,inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior)

n = power-law index

The Cross model is commonly used to describe the low-shear-rate behavior of the viscosity.

8.4.5.4.1. Inputs for the Cross Model

To use the Cross model, choose **cross** in the drop-down list to the right of **Viscosity**. The *Cross Model Dialog Box* (p. 1757) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Zero Shear Viscosity** η_0, **Time Constant** λ, **Power-Law Index**n, **Reference Temperature**T_α, and **Activation Energy/R**, which is the ratio of the activation energy to the thermodynamic constant α.

8.4.5.5. Herschel-Bulkley Model for Bingham Plastics

The power law model described above is valid for fluids for which the shear stress is zero when the strain rate is zero. Bingham plastics are characterized by a non-zero shear stress when the strain rate is zero.

$$\overline{\overline{\tau}} = \overline{\overline{\tau}}_0 + \eta\,\overline{\overline{D}} \tag{8–35}$$

where τ_0 is the yield stress:

- For $\tau < \tau_0$, the material remains rigid.
- For $\tau > \tau_0$, the material flows as a power-law fluid.

The Herschel-Bulkley model combines the effects of Bingham and power-law behavior in a fluid. For low strain rates ($\dot{\gamma} < \tau_0/\mu_0$), the "rigid" material acts like a very viscous fluid with viscosity μ_0. As the strain rate increases and the yield stress threshold, τ_0, is passed, the fluid behavior is described by a power law.

For $\dot{\gamma} > \dot{\gamma}_c$

$$\eta = \frac{\tau_0}{\dot{\gamma}} + k\left(\frac{\dot{\gamma}}{\dot{\gamma}_c}\right)^{n-1} \tag{8–36}$$

For $\dot{\gamma} < \dot{\gamma}_c$

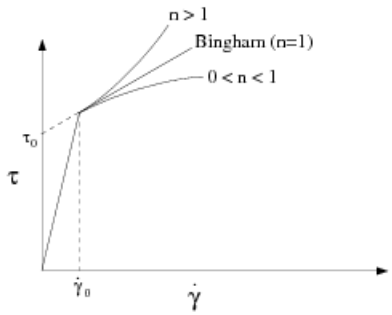
$$\eta = \tau_0 \frac{\left(2 - \dot{\gamma}/\dot{\gamma}_c\right)}{\dot{\gamma}_c} + k\left[\,(2-n) + (n-1)\frac{\dot{\gamma}}{\dot{\gamma}_c}\,\right] \tag{8–37}$$

where k is the consistency factor, and n is the power-law index.

Figure 8.16 (p. 418) shows how shear stress (τ) varies with shear rate ($\dot{\gamma}$) for the Herschel-Bulkley model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.16 Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model



If you choose the Herschel-Bulkley model for Bingham plastics, *Equation 8–36* (p. 417) will be used to determine the fluid viscosity.

The Herschel-Bulkley model is commonly used to describe materials such as concrete, mud, dough, and toothpaste, for which a constant viscosity after a critical shear stress is a reasonable assumption. In addition to the transition behavior between a flow and no-flow regime, the Herschel-Bulkley model can also exhibit a shear-thinning or shear-thickening behavior depending on the value of n.

8.4.5.5.1. Inputs for the Herschel-Bulkley Model

To use the Herschel-Bulkley model, choose **herschel-bulkley** in the drop-down list to the right of **Viscosity**. The *Herschel-Bulkley Dialog Box* (p. 1758) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Consistency Index**k, **Power-Law Index**n, **Yield Stress Threshold** τ_0, **Critical Shear Rate**$\dot{\gamma}_c$, **Reference Temperature** T_α, and the ratio of the activation energy to thermodynamic constant α, **Activation Energy/R**.

8.5. Thermal Conductivity

The thermal conductivity must be defined when heat transfer is active. You will need to define thermal conductivity when you are modeling energy and viscous flow.

ANSYS FLUENT provides several options for definition of the thermal conductivity:

- constant thermal conductivity
- temperature- and/or composition-dependent thermal conductivity
- kinetic theory
- anisotropic (anisotropic, biaxial, orthotropic, cylindrical orthotropic) (for solid materials only)
- user-defined

Thermal conductivity can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736).). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

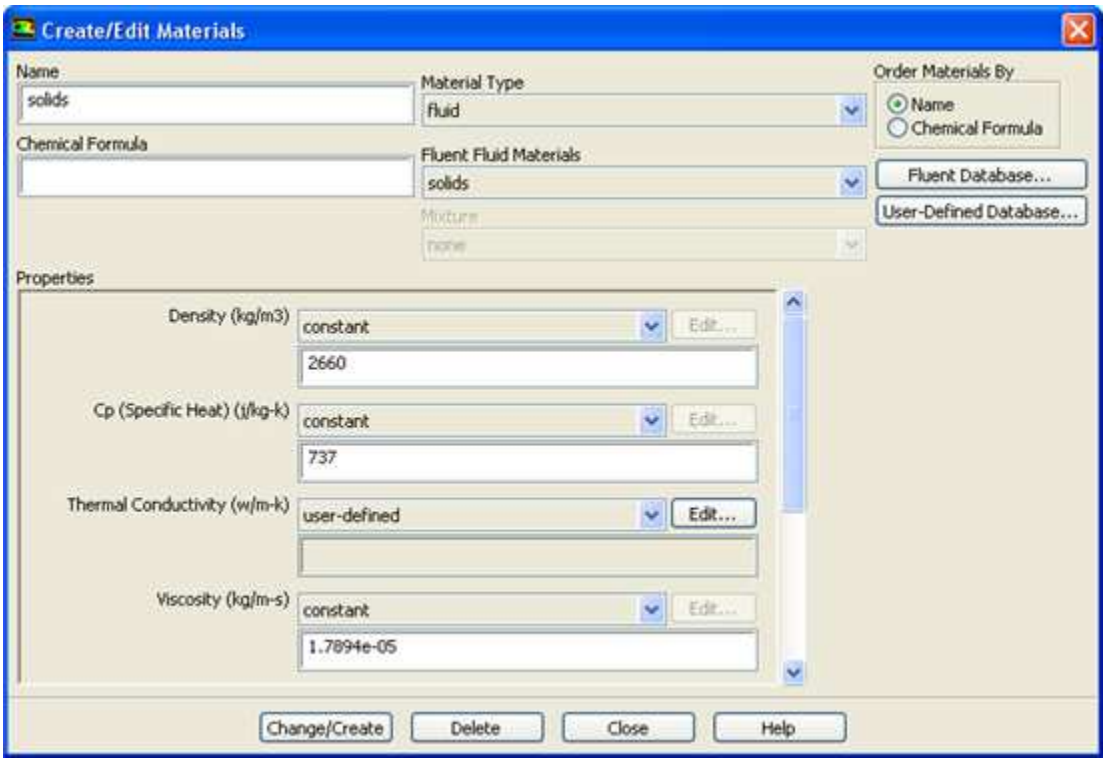
Each of these input options and the governing physical models are detailed in this section. User-defined functions (UDFs) are described in the UDF Manual.

In all cases, you will define the **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736) (*Figure 8.17* (p. 419)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials → Create/Edit...

Figure 8.17 The Create/Edit Materials Dialog Box



Thermal conductivity is defined in units of W/m-K in SI units or BTU/hr-ft-°R in British units.

For additional information, please see the following sections:

8.5.1. Constant Thermal Conductivity
8.5.2. Thermal Conductivity as a Function of Temperature
8.5.3. Thermal Conductivity Using Kinetic Theory
8.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures
8.5.5. Anisotropic Thermal Conductivity for Solids

8.5.1. Constant Thermal Conductivity

If you want to define the thermal conductivity as a constant , check that **constant** is selected in the drop-down list to the right of **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736) (*Figure 8.17* (p. 419)), and enter the value of thermal conductivity for the material.

For the default fluid (air), the thermal conductivity is 0.0242 W/m-K.

8.5.2. Thermal Conductivity as a Function of Temperature

You can also choose to define the thermal conductivity as a function of temperature. Three types of functions are available:

- piecewise-linear:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k(T) = k_n + \frac{k_{n+1} - k_n}{T_{n+1} - T_n}(T - T_n) \quad \textbf{(8–38)}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < \mathrm{T}_{max,1} : k(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < \mathrm{T}_{max,2} : k(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \quad \textbf{(8–39)}$$

- polynomial:

$$k(T) = A_1 + A_2 T + A_3 T^2 + \ldots \quad \textbf{(8–40)}$$

You can input the data pairs (T_n, k_n), ranges and coefficients A_i and B_i, or coefficients A_i that describe these functions using the *Create/Edit Materials Dialog Box* (p. 1736), as described in *Defining Properties Using Temperature-Dependent Functions* (p. 398).

8.5.3. Thermal Conductivity Using Kinetic Theory

If you are using the gas law (as described in *Density* (p. 403)), you have the option to define the thermal conductivity using kinetic theory as

$$k = \frac{15}{4}\frac{R}{M_w}\mu\left[\frac{4}{15}\frac{c_p M_w}{R} + \frac{1}{3}\right] \quad \textbf{(8–41)}$$

where R is the universal gas constant, M_w is the molecular weight, μ is the material's specified or computed viscosity, and c_p is the material's specified or computed specific heat capacity.

To enable the use of this equation for calculating thermal conductivity, select **kinetic-theory** from the drop-down list to the right of **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). The solver will use *Equation 8–41* (p. 420) to compute the thermal conductivity.

8.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent thermal conductivity. (Note that you can also define the thermal conductivity of the mixture as a constant value or a function of temperature, or using kinetic theory.)

To define a composition-dependent thermal conductivity for a mixture, follow these steps:

1. For the mixture material, choose **mass-weighted-mixing-law** or, if you are using the ideal gas law, **ideal-gas-mixing-law** in the drop-down list to the right of **Thermal Conductivity**. If you have a user-defined function that you want to use to model the thermal conductivity, you can choose either the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. Note that only the constant, the polynomial methods and the user-defined methods are available.

Important

If you use **ideal-gas-mixing-law** for the thermal conductivity of a mixture, you must use **ideal-gas-mixing-law** or **mass-weighted-mixing-law** for viscosity, because these two viscosity specification methods are the only ones that allow specification of the component viscosities, which are used in the ideal gas law for thermal conductivity (*Equation 8–42* (p. 421)).

2. Click **Change/Create**.
3. Define the thermal conductivity for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent thermal conductivities for the individual species. You may also use kinetic theory for the individual thermal conductivities, if applicable.
4. If you selected **user-defined-mixing-law**, define the thermal conductivity for each of the fluid materials that comprise the mixture. You may define constant, or (if applicable) temperature-dependent thermal conductivities, or user-defined thermal conductivities for the individual species. More information about defining properties with user-defined functions can be found in the UDF Manual.

If you are using the ideal gas law, the solver will compute the mixture thermal conductivity based on kinetic theory as

$$k = \sum_i \frac{X_i k_i}{\sum_j X_j \phi_{ij}} \tag{8–42}$$

where

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{1/2} \left(\frac{M_{w,j}}{M_{w,i}}\right)^{1/4}\right]^2}{\left[8\left(1 + \frac{M_{w,i}}{M_{w,j}}\right)\right]^{1/2}} \tag{8–43}$$

and X_i is the mole fraction of species i.

For non-ideal gases, the mixture thermal conductivity is computed based on a simple mass fraction average of the pure species conductivities:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k = \sum_i Y_i k_i \tag{8–44}$$

8.5.5. Anisotropic Thermal Conductivity for Solids

The anisotropic conductivity option in ANSYS FLUENT solves the conduction equation in solids with the thermal conductivity specified as a matrix. The heat flux vector is written as

$$q_i = -k_{ij} \frac{\partial T}{\partial x_j} \tag{8–45}$$

The following options are available for defining anisotropic thermal conductivity in ANSYS FLUENT. These are discussed below.

- anisotropic
- biaxial
- orthotropic
- cylindrical orthotropic

Important

Note that the anisotropic conductivity options are available only with the pressure-based solver; you cannot use them with the density-based solvers.

8.5.5.1. Anisotropic Thermal Conductivity

For anisotropic diffusion, the thermal conductivity matrix (*Equation 8–45* (p. 422)) is specified as

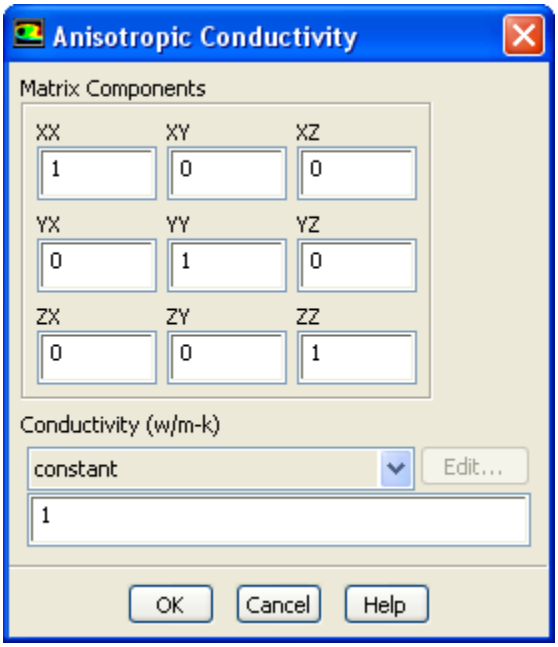
$$k_{ij} = k \hat{e}_{ij} \tag{8–46}$$

where k is the conductivity and $\hat{e}_{ij}$ is a matrix (2 $\times$ 2 for two dimensions and 3 $\times$ 3 for three-dimensional problems. Note that $\hat{e}_{ij}$ can be a non-symmetric matrix.

To define anisotropic thermal conductivity for a solid material, select **anisotropic** for **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736) (*Figure 8.17* (p. 419)). This will open the *Anisotropic Conductivity Dialog Box* (p. 1762) (*Figure 8.18* (p. 423)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.18 The Anisotropic Conductivity Dialog Box



In the *Anisotropic Conductivity Dialog Box* (p. 1762), enter the **Matrix Components** of matrix $\hat{e}_{ij}$ and then select the **Conductivity** (k in *Equation 8–46* (p. 422)) to be a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function. See *Constant Thermal Conductivity* (p. 419) and *Thermal Conductivity as a Function of Temperature* (p. 419) for details on constants and thermal polynomial functions.

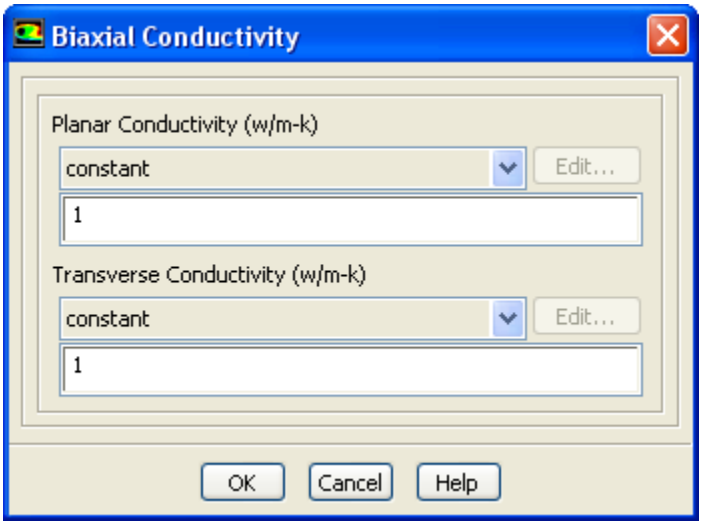
When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. Details about user-defined functions can be found in the UDF Manual.

8.5.5.2. Biaxial Thermal Conductivity

Biaxial thermal conductivity is mainly applicable to solid materials used for the wall shell conduction model. To define a biaxial thermal conductivity, select **biaxial** in the drop-down list for **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). This opens the *Biaxial Conductivity Dialog Box* (p. 1759) (*Figure 8.19* (p. 424)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.19 The Biaxial Conductivity Dialog Box



In the *Biaxial Conductivity Dialog Box* (p. 1759), both the conductivity normal to the surface of the solid region (**Transverse Conductivity**) and the conductivity within the shell or solid region (**Planar Conductivity**) can be defined as **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial**. See *Constant Thermal Conductivity* (p. 419) and *Thermal Conductivity as a Function of Temperature* (p. 419) for details on these parameters. Within the shell, however, the conductivity is isotropic. See *Figure 14.5* (p. 703)

8.5.5.3. Orthotropic Thermal Conductivity

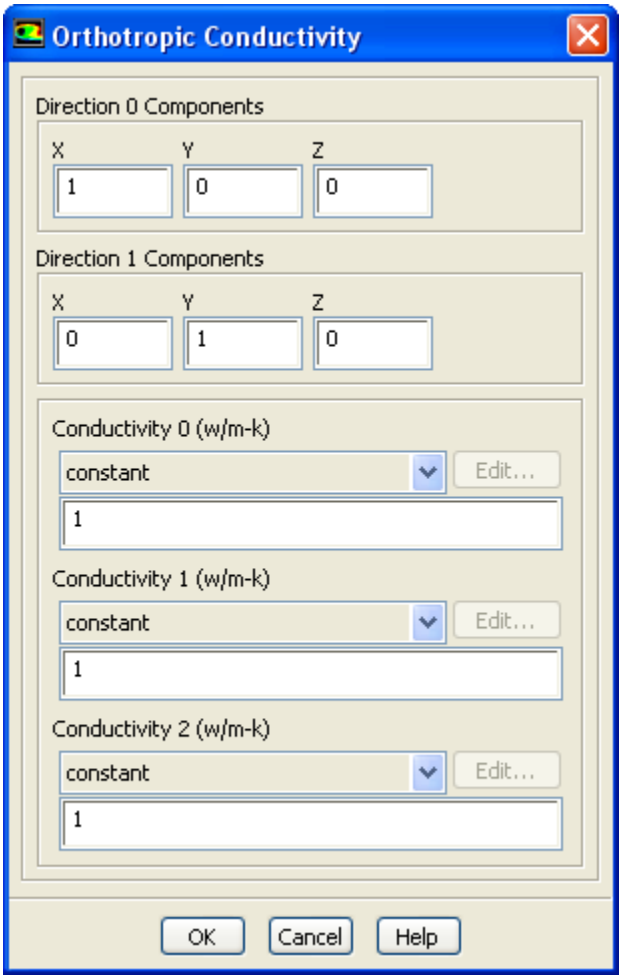
When the orthotropic thermal conductivity is used, the thermal conductivities $\left(k_\xi, k_\eta, k_\zeta\right)$ in the principal directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are specified. The conductivity matrix is then computed as

$$k_{ij} = k_\xi e_{\xi i} e_{\xi j} + k_\eta e_{\eta i} e_{\eta j} + k_\zeta e_{\zeta i} e_{\zeta j} \qquad \textbf{(8–47)}$$

To define an orthotropic thermal conductivity in solids, select **orthotropic** in the drop-down list for **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). This opens the *Orthotropic Conductivity Dialog Box* (p. 1761) (*Figure 8.20* (p. 425)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.20 The Orthotropic Conductivity Dialog Box



Since the directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are mutually orthogonal, only the first two need to be specified for three-dimensional problems. $\hat{e}_\xi$ is defined using **X, Y, Z** under **Direction 0 Components**, and $\hat{e}_\eta$ is defined using **X, Y, Z** under **Direction 1 Components**. You can define **Conductivity 0** $\left(k_\xi\right)$, **Conductivity 1** $\left(k_\eta\right)$, and **Conductivity 2** $\left(k_\zeta\right)$ as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial** functions of temperature, or **user-defined**. See *Constant Thermal Conductivity* (p. 419) and *Thermal Conductivity as a Function of Temperature* (p. 419) for details on constant and temperature profile functions.

When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

Important

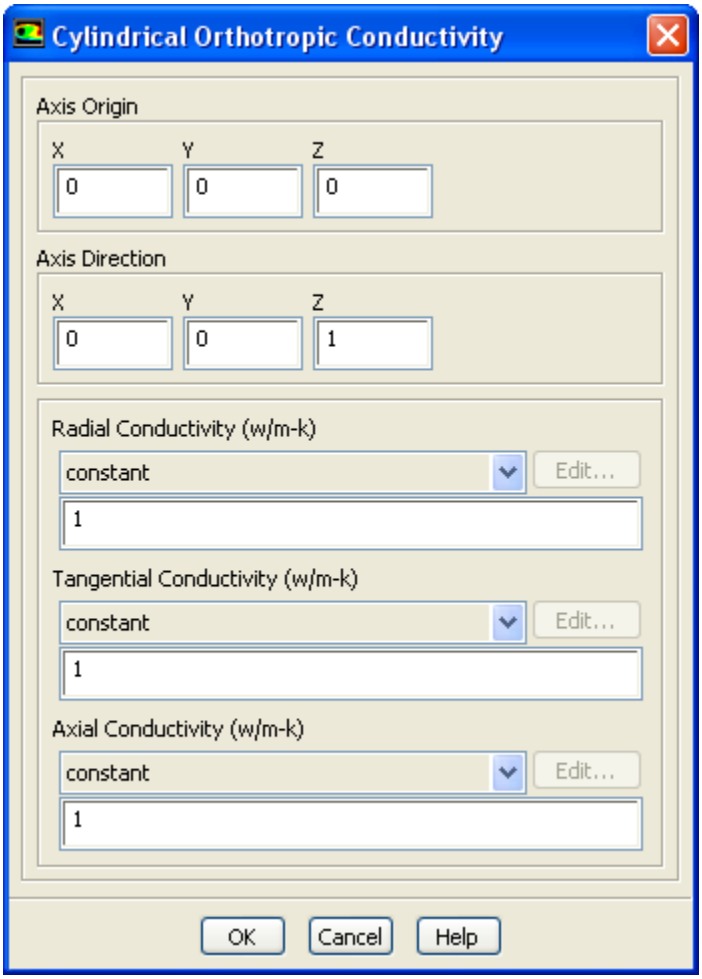
For two-dimensional problems, only the functions $\left(k_\xi, k_\eta\right)$ and the unit vector $\left(\hat{e}_\xi\right)$ need to be specified.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.5.5.4. Cylindrical Orthotropic Thermal Conductivity

The orthotropic conductivity of solids can be specified in cylindrical coordinates. To define the orthotropic thermal conductivity in cylindrical coordinates, select **cyl-orthotropic** in the drop-down list for **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). This opens the *Cylindrical Orthotropic Conductivity Dialog Box* (p. 1760) (*Figure 8.21* (p. 426)).

Figure 8.21 The Cylindrical Orthotropic Conductivity Dialog Box



In three-dimensional cases, the origin and the direction of the cylindrical coordinate system must be specified along with the radial, tangential, and axial direction conductivities. In two-dimensional cases, the origin of the cylindrical coordinate system must be specified along with the radial and tangential direction conductivities. Note that in two-dimensional cases, the direction is always along the +z axis. ANSYS FLUENT will automatically compute the anisotropic conductivity matrix at each cell from this input. The calculation is based on the location of the cell in the cylindrical coordinate system specified.

You can define the **Radial Conductivity**, **Tangential Conductivity**, and **Axial Conductivity** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or as user-defined functions of temperature. See *Constant Thermal Conductivity* (p. 419) and *Thermal Conductivity as a Function of Temperature* (p. 419) for details on constant and thermal profile functions.

When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or inter-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

preted the UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

Important

For conductivity calculations near the wall, the cell next to the wall is chosen for computing the conductivity matrix instead of the wall itself.

8.6. User-Defined Scalar (UDS) Diffusivity

There are two types of UDS diffusivity that you can specify in ANSYS FLUENT: isotropic and anisotropic. Diffusion is isotropic when it is the same in all directions. Isotropic diffusion coefficients can be specified in two ways: either as a single **user-defined** that applies to all UDS transport equations defined for your model; or on a per-scalar basis as **constants**, **polynomial** functions of temperature, or **user-defined** functions.

Diffusion is anisotropic when the diffusion coefficients are different in different directions. Anisotropic diffusion can be specified by a tensor diffusion coefficient matrix Γ (*Equation 8–48* (p. 427)) for each UDS (in both fluid and solid zones) in four different ways: general **anisotropic**, **orthotropic**, **cyl-orthotropic**, and **user-defined-anisotropic**. All UDS diffusivity parameters are set from the *Create/Edit Materials Dialog Box* (p. 1736) and are discussed below. Note that details about how to define and use UDFs in UDS transport equations is discussed in the UDF Manual.

The second-order diffusion term in the most general form is

$$\nabla \cdot \left(\Gamma \cdot \nabla \phi^k \right) \tag{8–48}$$

where Γ is a 3×3 tensor in 3D.

UDS diffusivity can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736).). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

For additional information, please see the following sections:

8.6.1. Isotropic Diffusion
8.6.2. Anisotropic Diffusion
8.6.3. User-Defined Anisotropic Diffusivity

8.6.1. Isotropic Diffusion

For isotropic diffusion, Γ in *Equation 8–48* (p. 427) is equal to a scalar Γ times the identity matrix and the equation reduces to

$$\nabla \cdot \left(\Gamma \nabla \phi^k \right) \tag{8–49}$$

You can specify isotropic diffusivity as a single user-defined function that applies to all UDS transport equations. For this case, choose **user-defined** from the drop-down list for **UDS Diffusivity** in the *Create/Edit Materials Dialog Box* (p. 1736).

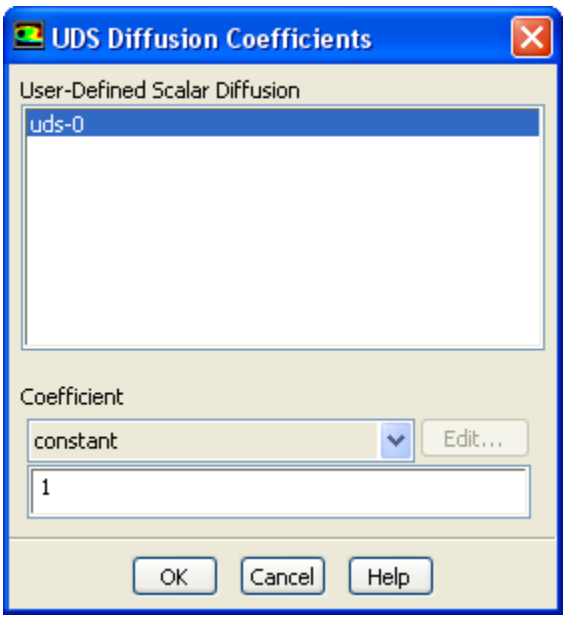
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials

If you have previously loaded a compiled UDF library or have interpreted the UDF, then the *User-Defined Functions Dialog Box* (p. 1753) will open, allowing you to hook the `DEFINE_DIFFUSIVITY` UDF to ANSYS FLUENT. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

Isotropic diffusion coefficients can also be defined on a per-scalar basis by selecting **defined-per-uds** from the drop-down list for **UDS Diffusivity** in the *Create/Edit Materials Dialog Box* (p. 1736). This will open the *UDS Diffusion Coefficients Dialog Box* (p. 1775) (*Figure 8.22* (p. 428)).

Figure 8.22 The UDS Diffusion Coefficients Dialog Box



In the *UDS Diffusion Coefficients Dialog Box* (p. 1775), select a scalar equation (e.g., **uds-0**) and then choose a **constant**, **polynomial**, or **user-defined** function from the **Coefficient** drop-down list. For the default fluid (air), the constant diffusion coefficient is 1 kg/m-s. If you choose **polynomial**, the *Polynomial Profile Dialog Box* (p. 1751) will open and you can specify your coefficients as a function of temperature. See *Inputs for Polynomial Functions* (p. 399) for details.

When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

8.6.2. Anisotropic Diffusion

You can specify anisotropic diffusion coefficients in both fluid and solid zones by defining the tensor diffusion coefficient matrix Γ (*Equation 8–48* (p. 427)) on a per-scalar basis. You can use anisotropic diffusivity for UDS scalar transport equations to model species transport equations in porous media and in solids where species diffusion shows anisotropic behavior.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- Note that the anisotropic diffusion options discussed in the following sections are available with the pressure-based solver and the density-based solvers.
- UDS diffusion coefficients can be postprocessed only in those cells which have isotropic diffusivity. In all other cells, the diffusion coefficient will be zero.

In all cases, you enable anisotropic diffusion by selecting **defined-per-uds** under **UDS Diffusivity** in the *Create/Edit Materials Dialog Box* (p. 1736). This will open the *UDS Diffusion Coefficients Dialog Box* (p. 1775) (*Figure 8.22* (p. 428)).

Materials

In the *UDS Diffusion Coefficients Dialog Box* (p. 1775), select a scalar equation (e.g., **uds-0**) and then choose one of the following methods under **Coefficient** to specify the anisotropic diffusion coefficient. These methods are described in detail below.

- anisotropic
- orthotropic
- cylindrical orthotropic
- user-defined anisotropic

8.6.2.1. Anisotropic Diffusivity

For anisotropic diffusivity, you can specify Γ in *Equation 8–48* (p. 427) in the form $K\Gamma$ where K is a constant $3x3$ matrix in3D and Γ is a scalar multiplier.

The diffusion coefficient matrix is specified as

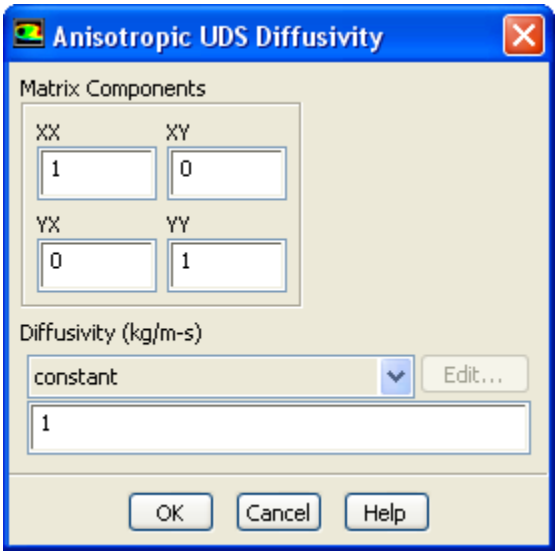
$$k_{ij}=k\hat{e}_{ij} \tag{8–50}$$

where k is the diffusivity and $\hat{e}_{ij}$ is a matrix (2 $\times$ 2 for two dimensions and 3 $\times$ 3 for three-dimensional problems). Note that $\hat{e}_{ij}$ can be a non-symmetric matrix.

To specify anisotropic diffusion coefficients, first select a scalar equation (e.g., **uds-0**) from the **User-Defined Scalar Diffusion** list in the *UDS Diffusion Coefficients Dialog Box* (p. 1775) (*Figure 8.22* (p. 428)). Then choose **anisotropic** in the drop-down list under **Coefficient**. This will open the **Anisotropic UDS Diffusivity** dialog box (*Figure 8.23* (p. 430)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.23 The Anisotropic UDS Diffusivity Dialog Box



In the **Anisotropic UDS Diffusivity** dialog box, enter the **Matrix Components** and then select the **Diffusivity** to be a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined**. See *Inputs for Polynomial Functions* (p. 399), *Inputs for Piecewise-Linear Functions* (p. 400), and *Inputs for Piecewise-Polynomial Functions* (p. 401) for details on polynomial temperature functions.

When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

8.6.2.2. Orthotropic Diffusivity

For orthotropic diffusivity, you can specify Γ in *Equation 8–48* (p. 427) through 'principal' direction vectors and diffusion coefficients along these directions. ANSYS FLUENT, in turn, computes Γ from parameters that you supply. The principal directions are the same everywhere, but each of he directional diffusion coefficients can be specified as a constant, polynomial function of temperature, or through user-defined functions.

When orthotropic diffusivity is used, the diffusion coefficients $\left(k_\xi, k_\eta, k_\zeta\right)$ in the principal directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are specified. The diffusivity matrix is then computed as

$$k_{ij} = k_\xi e_{\xi i} e_{\xi j} + k_\eta e_{\eta i} e_{\eta j} + k_\zeta e_{\zeta i} e_{\zeta j} \qquad \textbf{(8–51)}$$

Important

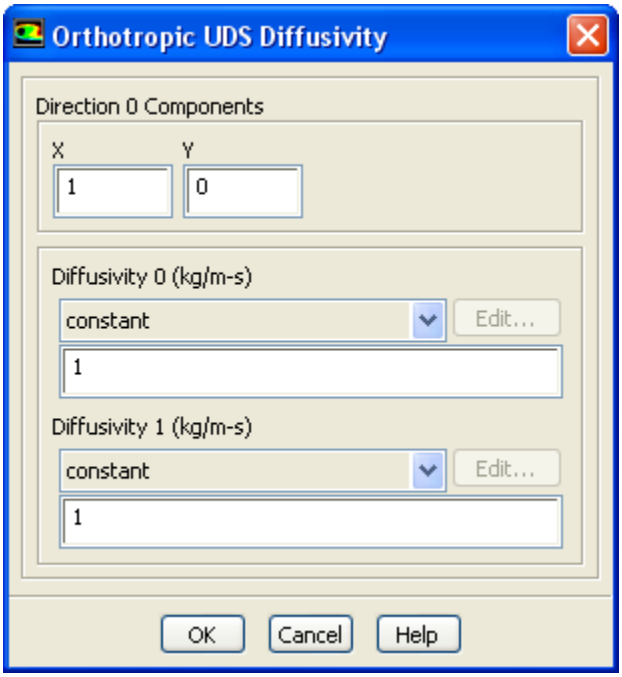
For two-dimensional problems, only the functions $\left(k_\xi, k_\eta\right)$ and the unit vector $\left(\hat{e}_\xi\right)$ need to be specified.

To specify orthotropic diffusion coefficients, first select a scalar equation (e.g., **uds-0**) from the **User-Defined Scalar Diffusion** list in the *UDS Diffusion Coefficients Dialog Box* (p. 1775) (*Figure 8.22* (p. 428)). Then choose

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

orthotropic in the drop-down list under **Coefficient**. This will open the **Orthotropic UDS Diffusivity** dialog box (*Figure 8.24* (p. 431)).

Figure 8.24 The Orthotropic UDS Diffusivity Dialog Box



Since the directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are mutually orthogonal, only the first two need to be specified for three-dimensional problems. $\hat{e}_\xi$ is defined using **X,Y,Z** under **Direction 0 Components**, and $\hat{e}_\eta$ is defined using **X,Y,Z** under **Direction 1 Components**. You can define **Diffusivity 0** $\left(k_\xi\right)$, **Diffusivity 1** $\left(k_\eta\right)$, and **Diffusivity 2** $\left(k_\zeta\right)$ as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial** functions of temperature, or **user-defined**. See *Inputs for Polynomial Functions* (p. 399), *Inputs for Piecewise-Linear Functions* (p. 400), and *Inputs for Piecewise-Polynomial Functions* (p. 401) for details on polynomial temperature functions.

When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

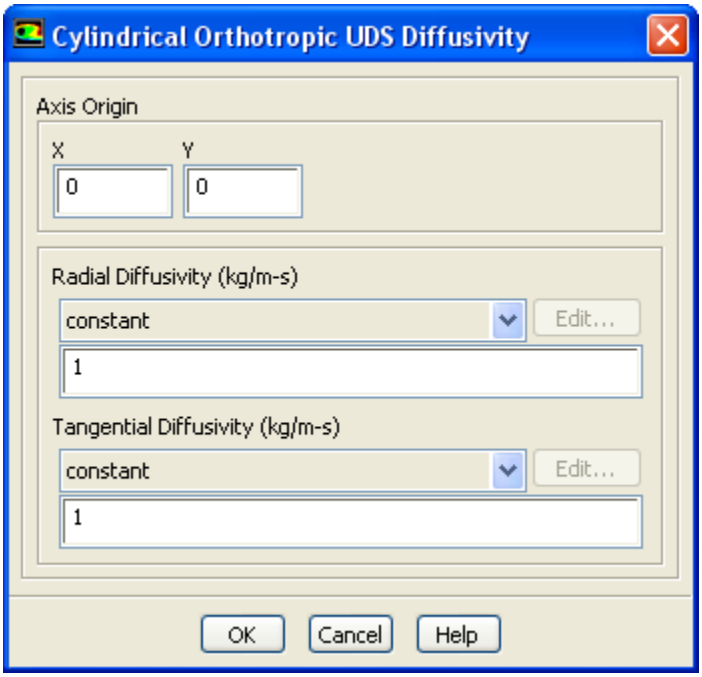
8.6.2.3. Cylindrical Orthotropic Diffusivity

Orthotropic UDS diffusivity can also be specified on a per-scalar basis in cylindrical coordinates. This method is similar to orthotropic UDS diffusivity, except that the principal directions are specified as radial, tangential, and axial.

To specify cylindrical orthotropic diffusion coefficients, first select a scalar equation (e.g., **uds-0**) from the **User-Defined Scalar Diffusion** list in the *UDS Diffusion Coefficients Dialog Box* (p. 1775) (*Figure 8.22* (p. 428)). Then choose **cyl-orthotropic** in the drop-down list under **Coefficient**. This will open the **Cylindrical Orthotropic UDS Diffusivity** dialog box (*Figure 8.25* (p. 432)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.25 The Cylindrical Orthotropic UDS Diffusivity Dialog Box



In three-dimensional cases, the origin and the direction of the cylindrical coordinate system must be specified along with the radial, tangential, and axial direction conductivities. In two-dimensional cases, the origin of the cylindrical coordinate system must be specified along with the radial and tangential direction conductivities. Note that in two-dimensional cases, the direction is always along the +z axis. ANSYS FLUENT will automatically compute the anisotropic diffusivity matrix at each cell from this input. The calculation is based on the location of the cell in the cylindrical coordinate system specified.

You can define the **Radial Diffusivity**, **Tangential Diffusivity**, and **Axial Diffusivity** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or as **user-defined** functions of temperature, using the drop-down list below each of the diffusivities. See *Inputs for Polynomial Functions* (p. 399), *Inputs for Piecewise-Linear Functions* (p. 400), and *Inputs for Piecewise-Polynomial Functions* (p. 401) for details on polynomial temperature functions.

When you select the **user-defined** option, the *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

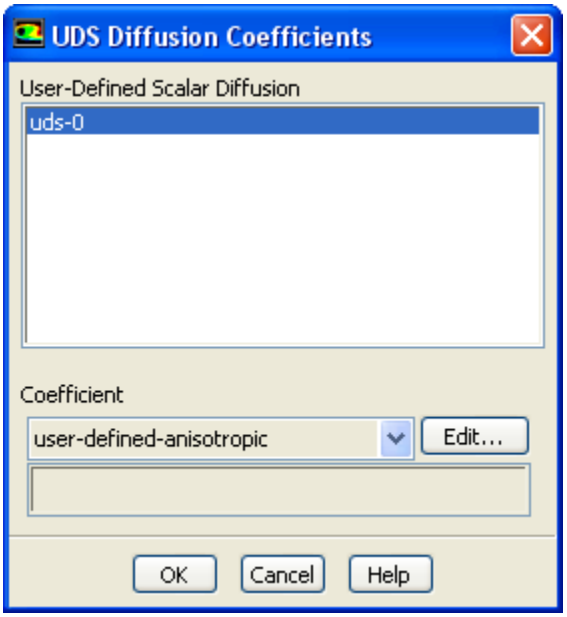
8.6.3. User-Defined Anisotropic Diffusivity

You can specify Γ in *Equation 8–48* (p. 427) on a per-scalar basis, directly, through user-defined functions (UDFs).

To specify a UDF for anisotropic diffusivity on a per-scalar basis, first select a scalar equation (e.g., **uds-0**) from the **User-Defined Scalar Diffusion** list in the *UDS Diffusion Coefficients Dialog Box* (p. 1775) (*Figure 8.26* (p. 433)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.26 The UDS Diffusion Coefficients Dialog Box



Then choose **user-defined-anisotropic** in the drop-down list under **Coefficient**. The *User-Defined Functions Dialog Box* (p. 1753) will open allowing you to hook a `DEFINE_ANISOTROPIC_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the UDF Manual.

8.7. Specific Heat Capacity

The specific heat capacity must be defined when the energy equation is active. ANSYS FLUENT provides several options for definition of the heat capacity:

- constant heat capacity
- temperature- and/or composition-dependent heat capacity
- kinetic theory

Each of these input options and the governing physical models are detailed in this section. In all cases, you will define the **Cp** in the *Create/Edit Materials Dialog Box* (p. 1736).

Materials

Specific heat capacity is input in units of J/kg-K in SI units or BTU/lbm-°R in British units.

Important

For combustion applications, a temperature-dependent specific heat is recommended.

Specific heat capacity can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736).). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, please see the following sections:

8.7.1. Input of Constant Specific Heat Capacity
8.7.2. Specific Heat Capacity as a Function of Temperature
8.7.3. Defining Specific Heat Capacity Using Kinetic Theory
8.7.4. Specific Heat Capacity as a Function of Composition

8.7.1. Input of Constant Specific Heat Capacity

If you want to define the heat capacity as a constant, check that **constant** is selected in the drop-down list to the right of **Cp** in the *Create/Edit Materials Dialog Box* (p. 1736), and enter the value of heat capacity.

The specific heat for the default fluid (air) is 1006.43 J/kg-K.

8.7.2. Specific Heat Capacity as a Function of Temperature

You can also choose to define the specific heat capacity as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$c_p(T) = c_{p_n} + \frac{c_{p_{n+1}} - c_{p_n}}{T_{n+1} - T_n}(T - T_n) \qquad \textbf{(8–52)}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < \mathrm{T}_{max,1} : c_p(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2} : c_p(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \qquad \textbf{(8–53)}$$

- polynomial:

$$c_p(T) = A_1 + A_2 T + A_3 T^2 + \ldots \qquad \textbf{(8–54)}$$

You can input the data pairs (T_n, c_{p_n}), ranges and coefficients A_i and B_i, or coefficients A_i that describe these functions using the *Create/Edit Materials Dialog Box* (p. 1736), as described in *Defining Properties Using Temperature-Dependent Functions* (p. 398).

8.7.3. Defining Specific Heat Capacity Using Kinetic Theory

If you are using the gas law (as described in *Density* (p. 403)), you have the option to define the specific heat capacity using kinetic theory as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$c_{p,i}=\frac{1}{2}\frac{R}{M_{w,i}}\left(f_i+2\right) \quad \textbf{(8–55)}$$

where f_i is the number of modes of energy storage (degrees of freedom) for the gas species i which you can input by selecting **kinetic-theory** from the drop-down list to the right of **Cp** in the *Create/Edit Materials Dialog Box* (p. 1736). The solver will use your kinetic theory inputs in *Equation 8–55* (p. 435) to compute the specific heat capacity. See *Kinetic Theory Parameters* (p. 448) for details about kinetic theory inputs.

8.7.4. Specific Heat Capacity as a Function of Composition

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent specific heat capacity. You can also define the heat capacity of the mixture as a constant value or a function of temperature, or using kinetic theory.

To define a composition-dependent specific heat capacity for a mixture, follow these steps:

1. For the mixture material, choose **mixing-law** in the drop-down list to the right of **Cp**.
2. Click **Change/Create**.
3. Define the specific heat capacity for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent heat capacities for the individual species. You may also use kinetic theory for the individual heat capacities, if applicable.

The solver will compute the mixture's specific heat capacity as a mass fraction average of the pure species heat capacities:

$$c_p=\sum_i Y_i c_{p,i} \quad \textbf{(8–56)}$$

8.8. Radiation Properties

When you have activated one of the radiation models (except for the surface-to-surface model, which requires no additional properties), there will be additional properties for you to set in the *Create/Edit Materials Dialog Box* (p. 1736):

- For the P-1 model, you will need to set the radiation **Absorption Coefficient** and **Scattering Coefficient** (a and σ_s in Equation 5–18 in the Theory Guide).
- For the Rosseland radiation model, you will also need to set the **Absorption Coefficient** and **Scattering Coefficient** (a and σ_s in Equation 5–19 in the Theory Guide).
- For the DTRM, only the **Absorption Coefficient** is required (a in Equation 5–52 in the Theory Guide).
- For the DO model set both, the **Absorption Coefficient** and the **Scattering Coefficient** (a and σ_s in Equation 5–59 in the Theory Guide). In addition, if you are modeling semi-transparent media, specify the **Refractive Index** (n_a or n_b in Equation 5–78 in the Theory Guide). With the DO model, you can specify radiation properties for solid materials, to be used when semi-transparent media are modeled.

For additional information, please see the following sections:

8.8.1. Absorption Coefficient
8.8.2. Scattering Coefficient
8.8.3. Refractive Index
8.8.4. Reporting the Radiation Properties

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.1. Absorption Coefficient

To define the absorption coefficient, you can specify a constant value, a temperature-dependent function (see *Defining Properties Using Temperature-Dependent Functions* (p. 398)), a composition-dependent function, or a user-defined function. The absorbing and emitting parts of the radiative transfer equation (RTE), Equation 5–17 in the Theory Guide, is a function of the absorption coefficient. The absorbing or emitting effects depend on the chosen radiation model. If there are only absorption effects, then Lambert's Law of absorption applies

$$I = I_o \exp(-ax) \tag{8–57}$$

where I is the radiation intensity, a is the absorption coefficient, and x is the distance through the material.

If you are modeling non-gray radiation with the P-1 or the DO radiation model, you also have the option to specify a constant absorption coefficient in each of the gray bands. The absorption coefficient is requested in units of 1/length. Along with the scattering coefficient, it describes the change in radiation intensity per unit length along the path through the fluid medium. Absorption coefficients can be computed using tables of emissivity for CO_2 and H_2O, which are generally available in textbooks on radiation heat transfer.

8.8.1.1. Inputs for a Constant Absorption Coefficient

To define a constant absorption coefficient, simply enter the value in the field next to **Absorption Coefficient** in the *Create/Edit Materials Dialog Box* (p. 1736). Select **constant** in the drop-down list first if it is not already selected.

8.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient

ANSYS FLUENT also allows you to input a composition-dependent absorption coefficient, where the local value of a is a function of the local mass fractions of water vapor and carbon dioxide. This modeling option can be useful for the simulation of radiation in combustion applications. The variable-absorption-coefficient model used by ANSYS FLUENT is the weighted-sum-of-gray-gases model (WSGGM) described in Radiation in Combusting Flows in the Theory Guide. To activate it, first enable the species calculation and make sure that CO_2 and H_2O are present in the mixture. Next, select **wsggm-domain-based**, **wsggm-user-specified** or **user-defined-wsggm** in the drop-down list to the right of **Absorption Coefficient** in the **Create/Edit Materials** dialog box. If you select **user-defined-wsggm**, the **User-Defined Functions** dialog box will open, allowing you to select a previously loaded compiled UDF library or a previously interpreted UDF (see Hooking DEFINE_WSGGM_ABS_COEFF UDFs in the UDF Manual). The WSGGM options differ in the method used to compute the path length, as described in the section that follows.

8.8.1.2.1. Path Length Inputs

When the WSGGM is used to compute the absorption coefficient, you will have a choice of methods used to calculate the path length s in Equation 5–101 in the Theory Guide. See Radiation in Combusting Flows in the Theory Guide to determine which method is appropriate for your case.

You will select the path length method when you choose the property input method for **Absorption Coefficient** as described previously.

- If you choose **wsggm-domain-based**, the mean-beam-length approach will be used for the calculation of a and ANSYS FLUENT will compute the mean beam length based on an average dimension of the domain; no further inputs are required.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you choose **wsggm-user-specified**, the mean-beam-length approach will be used, but you will set the mean beam length yourself in the **Path Length** field in the *Create/Edit Materials Dialog Box* (p. 1736). This dialog box will open when you choose **wsggm-user-specified**, and since it is a modal dialog box, you must tend to it immediately.
- If you choose **user-defined-wsggm**, ANSYS FLUENT will initially compute the absorption coefficient in the same manner as described for the **wsggm-domain-based** option; however, you have the option of writing a user-defined function that customizes this calculated value. If the soot model is enabled, you can also use the UDF to customize the soot absorption coefficient computed by ANSYS FLUENT. See DEFINE_WSGGM_ABS_COEFF in the UDF Manual for further details.

8.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient

If you are using the non-gray DO model (see The DO Model Equations of the Theory Guide and *Defining Non-Gray Radiation for the DO Model* (p. 721)) or the non-gray P-1 model (see The P-1 Model Equations of the Theory Guide and *Setting Up the P-1 Model with Non-Gray Radiation* (p. 706)), you can specify a different constant absorption coefficient for each of the bands used by the gray-band model. Select **gray-band** from the **Absorption Coefficient** drop-down list in the **Create/Edit Materials** dialog box and then define the absorption coefficient for each band in the *Gray-Band Absorption Coefficient Dialog Box* (p. 1776). (Note that because this is a modal dialog box, you must tend to it immediately.)

8.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient

ANSYS FLUENT will include the effect of particles on the absorption coefficient if you have turned on the **Particle Radiation Interaction** option in the *Discrete Phase Model Dialog Box* (p. 1723) (only for the P-1 and DO radiation models).

If you are modeling soot formation and you want to include the effect of soot formation on the absorption coefficient, turn on the **Soot-Radiation Interaction** in the *Soot Model Dialog Box* (p. 1717). The soot effects can be included for any of the radiation models, as long as you are using the WSGGM to compute a composition-dependent absorption coefficient. Note that you can use the **user-defined-wsggm** option to customize the soot absorption coefficient calculated by ANSYS FLUENT, as described previously.

8.8.2. Scattering Coefficient

The scattering coefficient is, by default, set to zero, and it is assumed to be isotropic. You can specify a constant value, a temperature-dependent function (see *Defining Properties Using Temperature-Dependent Functions* (p. 398)), or a user-defined function. You can also specify a non-isotropic phase function.

The scattering coefficient is requested in units of 1/length. Along with the absorption coefficient, it describes the change in radiation intensity per unit length along the path through the fluid medium. You may wish to increase the scattering coefficient in combustion systems, where particulates may be present.

8.8.2.1. Inputs for a Constant Scattering Coefficient

To define a constant scattering coefficient, simply enter the value in the field next to **Scattering Coefficient** in the *Create/Edit Materials Dialog Box* (p. 1736). (Select **constant** in the drop-down list first if it is not already selected.)

8.8.2.2. Inputs for the Scattering Phase Function

Scattering is assumed to be isotropic, by default, but you can also specify a linear-anisotropic scattering function. If you are using the DO model, Delta-Eddington and user-defined scattering functions are also available.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.2.2.1. Isotropic Phase Function

To model isotropic scattering, select **isotropic** in the **Scattering Phase Function** drop-down list. No further inputs are necessary. This is the default setting in ANSYS FLUENT.

8.8.2.2.2. Linear-Anisotropic Phase Function

To model anisotropic scattering, select **linear-anisotropic** in the **Scattering Phase Function** drop-down list and set the value of the phase function coefficient (C in Equation 5–19 in the Theory Guide).

8.8.2.2.3. Delta-Eddington Phase Function

To use a Delta-Eddington phase function, select **delta-eddington** in the **Scattering Phase Function** drop-down list. This will open the *Delta-Eddington Scattering Function Dialog Box* (p. 1777), in which you can specify the **Forward Scattering Factor** and **Asymmetry Factor** (f and C in Equation 5–68 in the Theory Guide). Since this is a modal dialog box, you must tend to it immediately. **Asymmetry Factor** (f and C in Equation 5–68 in the Theory Guide).

8.8.2.2.4. User-Defined Phase Function

To use a user-defined phase function, select **user-defined** in the **Scattering Phase Function** drop-down list. The user-defined function will contain specifications for Φ^* and f in Equation 5–69 in the Theory Guide. More information about user-defined functions can be found in the UDF Manual.

8.8.3. Refractive Index

The refractive index is the ratio of speed of light in the medium to the speed of light in vacuum. It is by default set to 1. You can specify a constant value in the field next to **Refractive Index**.

If you are using the non-gray DO model (see The DO Model Equations of the Theory Guide and *Defining Non-Gray Radiation for the DO Model* (p. 721)) or the non-gray P-1 model (see The P-1 Model Equations of the Theory Guide and *Setting Up the P-1 Model with Non-Gray Radiation* (p. 706)), you can specify a different constant refractive index for each of the bands used by the gray-band model. Select **refractive-band** from the **Refractive Index** drop-down list in the **Create/Edit Materials** dialog box and then define the refractive index for each band in the *Gray-Band Refractive Index Dialog Box* (p. 1777). Note that because this is a modal dialog box, you must tend to it immediately.

8.8.4. Reporting the Radiation Properties

You can display the computed local values for a and σ_s using the **Absorption Coefficient** and **Scattering Coefficient** items in the **Radiation...** category of the variable selection drop-down list that appears in post-processing dialog boxes. You will also find the **Refractive Index** in the **Radiation...** category.

8.9. Mass Diffusion Coefficients

For species transport calculations, there are two ways to model the diffusion of chemical species. For most applications the Fick's law approximation is adequate, but for some applications (e.g., diffusion-dominated laminar flows such as chemical vapor deposition), the full multicomponent diffusion model is recommended.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The full multicomponent diffusion model is enabled in the *Species Model Dialog Box* (p. 1681) and is computationally expensive.

For additional information, please see the following sections:

8.9.1. Fickian Diffusion
8.9.2. Full Multicomponent Diffusion
8.9.3. Thermal Diffusion Coefficients
8.9.4. Mass Diffusion Coefficient Inputs
8.9.5. Mass Diffusion Coefficient Inputs for Turbulent Flow

8.9.1. Fickian Diffusion

Mass diffusion coefficients are required whenever you are solving species transport equations in multicomponent flows. Mass diffusion coefficients are used to compute the diffusion flux of a chemical species in a laminar flow using (by default) Fick's law:

$$J_i = -\rho D_{i,m} \nabla Y_i - D_{T,i} \frac{\nabla T}{T} \tag{8–58}$$

where $D_{i,m}$ is the mass diffusion coefficient for species i in the mixture and $D_{T,i}$ is the thermal (Soret) diffusion coefficient.

Equation 8–58 (p. 439) is strictly valid when the mixture composition is not changing, or when $D_{i,m}$ is independent of composition. This is an acceptable approximation in dilute mixtures when $Y_i << 1$, for all i except the carrier gas. ANSYS FLUENT can also compute the transport of non-dilute mixtures in laminar flows by treating such mixtures as multicomponent systems. Within ANSYS FLUENT, $D_{i,m}$ can be specified in a variety of ways, including by specifying $_{ij}$, the binary mass diffusion coefficient of component i in component j. $_{ij}$ is not used directly, however; instead, the diffusion coefficient in the mixture, $D_{i,m}$, is computed as

$$D_{i,m} = \frac{1 - X_i}{\sum_{j, j \neq i} \left(X_j / {}_{ij} \right)} \tag{8–59}$$

where X_i is the mole fraction of species i. You can input $D_{i,m}$ or $_{ij}$ for each chemical species, as described in *Mass Diffusion Coefficient Inputs* (p. 442).

In turbulent flows, *Equation 8–58* (p. 439) is replaced with the following form:

$$J_i = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t} \right) \nabla Y_i - D_{T,i} \frac{\nabla T}{T} \tag{8–60}$$

where Sc_t is the effective Schmidt number for the turbulent flow:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Sc_t = \frac{\mu_t}{\rho D_t} \tag{8–61}$$

and D_t is the effective mass diffusion coefficient due to turbulence.

In turbulent flows your mass diffusion coefficient inputs consist of defining the molecular contribution to diffusion $D_{i,m}$ using the same methods available for the laminar case, with the added option to alter the default settings for the turbulent Schmidt number. As seen from *Equation 8–61* (p. 440), this parameter relates the effective mass diffusion coefficient due to turbulence with the eddy viscosity μ_t. As discussed in *Mass Diffusion Coefficient Inputs for Turbulent Flow* (p. 446), the turbulent diffusion coefficient normally overwhelms the laminar diffusion coefficient, so the default constant value for the laminar diffusion coefficient is usually acceptable.

8.9.2. Full Multicomponent Diffusion

A careful treatment of chemical species diffusion in the species transport and energy equations is important when details of the molecular transport processes are significant (e.g., in diffusion-dominated laminar flows). As one of the laminar-flow diffusion models, ANSYS FLUENT has the ability to model full multicomponent species transport.

8.9.2.1. General Theory

For multicomponent systems it is not possible, in general, to derive relations for the diffusion fluxes containing the gradient of only one component (as described in *Fickian Diffusion* (p. 439)). Here, the Maxwell-Stefan equations will be used to obtain the diffusive mass flux. This will lead to the definition of generalized Fick's law diffusion coefficients [*93*] (p. 2213). This method is preferred over computing the multicomponent diffusion coefficients since their evaluation requires the computation of N^2 co-factor determinants of size $(N-1)\times(N-1)$, and one determinant of size $N \times N$ [*87*] (p. 2213), where N is the number of chemical species.

8.9.2.2. Maxwell-Stefan Equations

From Merk [*55*] (p. 2212), the Maxwell-Stefan equations can be written as

$$\sum_{\substack{j=1\\ j\neq i}}^{N} \frac{X_i X_j}{\mathcal{D}_{ij}}\left(\vec{V}_j - \vec{V}_i\right) = \vec{d}_i - \frac{\nabla T}{T}\sum_{\substack{j=1\\ j\neq i}}^{N} \frac{X_i X_j}{\mathcal{D}_{ij}}\left(\frac{D_{T,j}}{\rho_j} - \frac{D_{T,i}}{\rho_i}\right) \tag{8–62}$$

where,

X = the mole fraction

$\vec{V}$ = the diffusion velocity

$\mathcal{D}_{ij}$ = the binary mass diffusion coefficient of specie i in specie j

D_T = the thermal diffusion coefficient.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For an ideal gas the Maxwell diffusion coefficients are equal to the binary diffusion coefficients. If the external force is assumed to be the same on all species and that pressure diffusion is negligible, then $\vec{d}_i = \nabla X_i$. Since the diffusive mass flux vector is $\vec{J}_i = \rho_i \vec{V}_i$, the above equation can be written as

$$\sum_{\substack{j=1 \\ j \neq i}}^{N} \frac{X_i X_j}{\mathcal{D}_{ij}} \left(\frac{\vec{J}_j}{\rho_j} - \frac{\vec{J}_i}{\rho_i} \right) = \nabla X_i - \frac{\nabla T}{T} \sum_{\substack{j=1 \\ j \neq i}}^{N} \frac{X_i X_j}{\mathcal{D}_{ij}} \left(\frac{D_{T,j}}{\rho_j} - \frac{D_{T,i}}{\rho_i} \right) \tag{8–63}$$

After some mathematical manipulations, the diffusive mass flux vector, $\vec{J}_i$, can be obtained from

$$\vec{J}_i = -\sum_{j=1}^{N-1} \rho D_{ij} \nabla Y_j - D_{T,i} \frac{\nabla T}{T} \tag{8–64}$$

where Y_j is the mass fraction of species j. Other terms are defined as follows:

$$\begin{aligned}
D_{ij} &= [D] = [A]^{-1}[B] \\
M_{w,m} &= \left(\sum_{i=1}^{N} \frac{Y_i}{M_{w,i}} \right) \\
A_{ii} &= -\left(\frac{X_i}{\mathcal{D}_{iN}} \frac{M_{w,m}}{M_{w,N}} + \sum_{\substack{j=1 \\ j \neq i}}^{N} \frac{X_j}{\mathcal{D}_{ij}} \frac{M_{w,m}}{M_{w,i}} \right) \\
A_{ij} &= X_i \left(\frac{1}{\mathcal{D}_{ij}} \frac{M_{w,m}}{M_{w,j}} - \frac{1}{\mathcal{D}_{iN}} \frac{M_{w,m}}{M_{w,N}} \right) \\
B_{ii} &= -\left(X_i \frac{M_{w,m}}{M_{w,N}} + (1 - X_i) \frac{M_{w,m}}{M_{w,i}} \right) \\
B_{ij} &= X_i \left(\frac{M_{w,m}}{M_{w,j}} - \frac{M_{w,m}}{M_{w,N}} \right)
\end{aligned} \tag{8–65}$$

where,

$[A]$ and $[B]$ = $(N-1) \times (N-1)$ matrices

$[D]$ = an $(N-1) \times (N-1)$ matrix of the generalized Fick's law diffusion coefficients D_{ij} [*93*]

M_w = the molecular weight

m = a subscript for the mixture

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.9.3. Thermal Diffusion Coefficients

The thermal diffusion coefficients can be defined as constants, polynomial functions, user-defined functions, or using the following empirically-based composition-dependent expression derived from [42] (p. 2211):

$$D_{T,i} = -2.59\times10^{-7}T^{0.659}\left[\frac{M_{w,i}^{0.511}X_i}{\sum_{i=1}^{N}M_{w,i}^{0.511}X_i} - Y_i\right]\cdot\left[\frac{\sum_{i=1}^{N}M_{w,i}^{0.511}X_i}{\sum_{i=1}^{N}M_{w,i}^{0.489}X_i}\right] \quad \textbf{(8–66)}$$

This form of the Soret diffusion coefficient will cause heavy molecules to diffuse less rapidly, and light molecules to diffuse more rapidly, towards heated surfaces.

8.9.4. Mass Diffusion Coefficient Inputs

By default, the solver computes the species diffusion using *Equation 8–58* (p. 439) (for laminar flows) with your inputs for $D_{i,m}$, the diffusion coefficient for species i in the mixture. For turbulent flows, species diffusion is computed with *Equation 8–60* (p. 439).

You can input the mass diffusion coefficients using one of the following methods:

- Constant dilute approximation (Fickian diffusion only): define one constant for all $D_{i,m}$.
- Dilute approximation (Fickian diffusion only): define each $D_{i,m}$ as a constant or as a polynomial function of temperature (if heat transfer is enabled).
- Multicomponent method: define the binary diffusion of species i in each species j, $_{ij}$ as a constant or a polynomial function of temperature, or (for ideal gases only) using kinetic theory.
- User-defined function (UDF): define a single function that will apply to all mass diffusion coefficients. This is done using the `DEFINE_DIFFUSIVITY` macro and is explained in the UDF Manual.

You should choose to input $D_{i,m}$ (using one of the first two methods) if you are modeling a dilute mixture, with chemical species present at low mass fraction in a "carrier" fluid that is present at high concentration. You may wish to define the individual binary mass diffusion coefficients, $_{ij}$, if you are modeling a non-dilute mixture. If you choose to define $_{ij}$, the solver will compute the diffusion of species i in the mixture using *Equation 8–59* (p. 439), unless you have enabled full multicomponent diffusion.

Important

If you want to use the full multicomponent diffusion model described in *Full Multicomponent Diffusion* (p. 440), turn on the **Full Multicomponent Diffusion** option in the *Species Model Dialog Box* (p. 1681), and then select the multicomponent diffusion model.

You will define $D_{i,m}$ or $_{ij}$ for each chemical species using the *Create/Edit Materials Dialog Box* (p. 1736).

Materials

The diffusion coefficients have units of m^2/s in SI units or ft^2/s in British units.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass diffusion can also be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list in the *Create/Edit Materials Dialog Box* (p. 1736). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

8.9.4.1. Constant Dilute Approximation Inputs

To use the constant dilute approximation method, follow these steps:

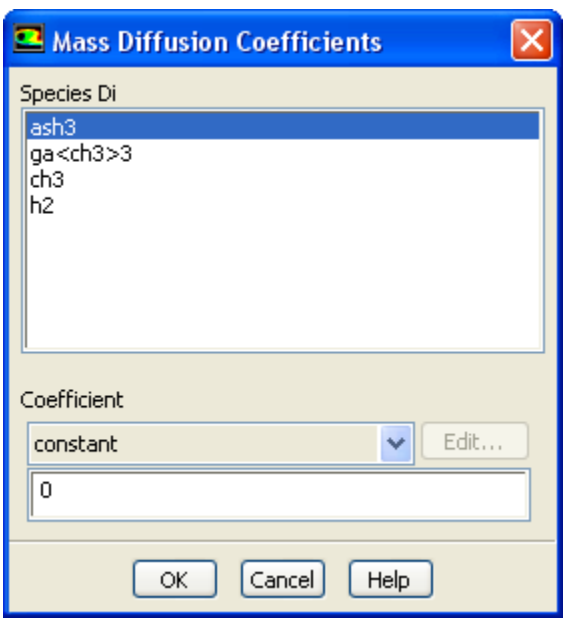
1. Select **constant-dilute-appx** in the drop-down list to the right of **Mass Diffusivity**.
2. Enter a single value of $D_{i,m}$. The same value will be used for the diffusion coefficient of each species in the mixture.

8.9.4.2. Dilute Approximation Inputs

To use the dilute approximation method, follow the steps below:

1. Select **dilute-approx** in the drop-down list to the right of **Mass Diffusivity**.
2. In the resulting *Mass Diffusion Coefficients Dialog Box* (p. 1773) (*Figure 8.27* (p. 443)), select the species in the **Species Di** list for which you are going to define the mass diffusion coefficient.

Figure 8.27 The Mass Diffusion Coefficients Dialog Box for Dilute Approximation



3. You can define $D_{i,m}$ for the selected species either as a constant value or (if heat transfer is active) as a polynomial function of temperature:
   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.
   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in *Inputs for Polynomial Functions* (p. 399).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$D_{i,m}=A_1+A_2T+A_3T^2+\ldots \quad \textbf{(8–67)}$$

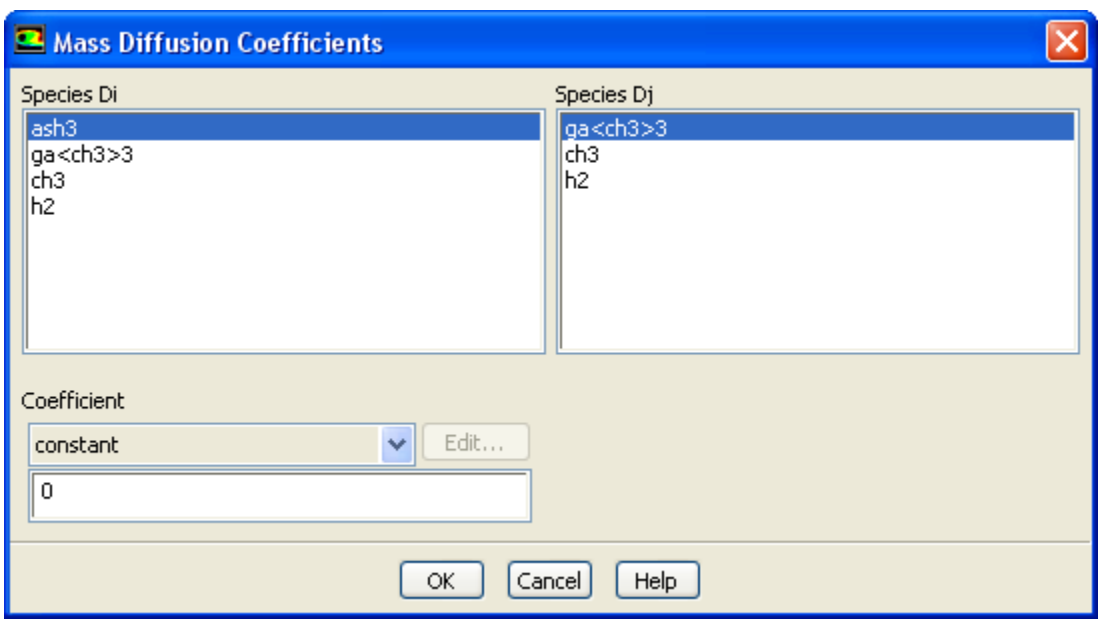
4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all species in the **Species Di** list in the *Mass Diffusion Coefficients Dialog Box* (p. 1773).

8.9.4.3. Multicomponent Method Inputs

To use the multicomponent method, and define constant or temperature-dependent diffusion coefficients, follow the steps below:

1. Select **multicomponent** in the drop-down list to the right of **Mass Diffusivity**.
2. In the resulting *Mass Diffusion Coefficients Dialog Box* (p. 1773) (*Figure 8.28* (p. 444)), select the species in the **Species Di** list and the **Species Dj** list for which you are going to define the mass diffusion coefficient D_{ij} for species i in species j.

Figure 8.28 The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method



3. You can define D_{ij} for the selected pair of species as a constant value or as a polynomial function of temperature (if heat transfer is active).
   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.
   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in *Inputs for Polynomial Functions* (p. 399).

$$D_{ij}=A_1+A_2T+A_3T^2+\ldots \quad \textbf{(8–68)}$$

4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all pairs of species in the **Species Di** and **Species Dj** lists in the *Mass Diffusion Coefficients Dialog Box* (p. 1773).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use the multicomponent method, and define the diffusion coefficient using kinetic theory (available only when the ideal gas law is used), follow these steps:

1. Choose **kinetic-theory** in the drop-down list to the right of **Mass Diffusivity**.
2. Click **Change/Create** after completing other property definitions for the mixture material.
3. Define the Lennard-Jones parameters, σ_i and $(\varepsilon/k_B)_i$, for each species (fluid material), as described in *Kinetic Theory Parameters* (p. 448).

The solver will use a modification of the Chapman-Enskog formula [*52*] (p. 2211) to compute the diffusion coefficient using kinetic theory:

$$D_{ij}=0.00188\frac{\left[T^3\left(\frac{1}{M_{w,i}}+\frac{1}{M_{w,j}}\right)\right]^{1/2}}{p_{abs}\sigma_{ij}^2\Omega_D} \tag{8–69}$$

where,

p_{abs} = the absolute pressure

Ω_D is the diffusion collision integral, which is a measure of the interaction of the molecules in the system

Ω_D is a function of the quantity T_D^*, where

$$T_D^*=\frac{T}{(\varepsilon/k_B)_{ij}} \tag{8–70}$$

k_B is the Boltzmann constant, which is defined as the gas constant, R, divided by Avogadro's number. $(\varepsilon/k_B)_{ij}$ for the mixture is the *geometric* average:

$$(\varepsilon/k_B)_{ij}=\sqrt{(\varepsilon/k_B)_i(\varepsilon/k_B)_j} \tag{8–71}$$

For a binary mixture, σ_{ij} is calculated as the *arithmetic* average of the individual σs:

$$\sigma_{ij}=\frac{1}{2}\left(\sigma_i+\sigma_j\right) \tag{8–72}$$

8.9.4.4. Thermal Diffusion Coefficient Inputs

If you have enabled thermal diffusion (in the *Species Model Dialog Box* (p. 1681)), you can define the thermal diffusion coefficients in the *Create/Edit Materials Dialog Box* (p. 1736) as follows:

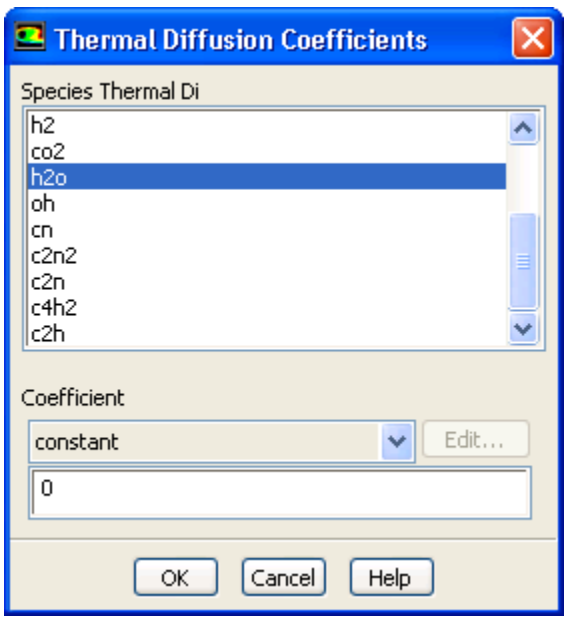
1. Select one of the following three methods in the drop-down list to the right of **Thermal Diffusion Coefficient**:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Choose **kinetic-theory** to have ANSYS FLUENT compute the thermal diffusion coefficients using the empirically-based expression in *Equation 8–66* (p. 442). No further inputs are required for this option.
- Choose **specified** to input the coefficient for each species. The *Thermal Diffusion Coefficients Dialog Box* (p. 1774) (*Figure 8.29* (p. 446)) will open. Further inputs are described in the step 2.
- Choose **user-defined** to use a user-defined function. More information about user-defined functions can be found in the UDF Manual.

2. If you choose **specified**, select the species in the **Species Thermal Di** list for which you are going to define the thermal diffusion coefficient.

Figure 8.29 The Thermal Diffusion Coefficients Dialog Box



3. Define $D_{T,i}$ for the selected species either as a constant value or as a polynomial function of temperature:
   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.
   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in *Defining Properties Using Temperature-Dependent Functions* (p. 398).
4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all species in the **Species Thermal Di** list in the *Thermal Diffusion Coefficients Dialog Box* (p. 1774).

8.9.5. Mass Diffusion Coefficient Inputs for Turbulent Flow

When your flow is turbulent, you will define $D_{i,m}$ or D_{ij}, as described for laminar flows in *Mass Diffusion Coefficient Inputs* (p. 442), and you will also have the option to alter the default setting for the turbulent Schmidt number, Sc_t, as defined in *Equation 8–61* (p. 440).

Usually, in a turbulent flow, the mass diffusion is dominated by the turbulent transport as determined by the turbulent Schmidt number (*Equation 8–61* (p. 440)). The turbulent Schmidt number measures the relative diffusion of momentum and mass due to turbulence and is on the order of unity in all turbulent flows. Because

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the turbulent Schmidt number is an empirical constant that is relatively insensitive to the molecular fluid properties, you will have little reason to alter the default value (0.7) for any species.

Should you wish to modify the Schmidt number, enter a new value for **Turb. Schmidt Number** in the *Viscous Model Dialog Box* (p. 1645).

Models → **Viscous** → **Edit...**

Important

Note that the full multicomponent diffusion model described in *Full Multicomponent Diffusion* (p. 440) is not recommended for turbulent flows.

8.10. Standard State Enthalpies

When you are solving a reacting flow using the finite-rate or eddy dissipation model, you will need to define the standard state enthalpy (also known as the formation enthalpy or heat of formation), h_j^0 for each species j. These inputs are used to define the mixture enthalpy as

$$H = \sum_j m_j \left[h_j^0 + \int_{T_{ref,j}}^{T} c_{p,j} dT \right] \tag{8–73}$$

where $T_{ref,j}$ is the reference temperature at which h_j^0 is defined. Standard state enthalpies are input in units of J/kg mol in SI units or in units of Btu/lb_mmol in British units.

For each species involved in the reaction (i.e., each fluid material contained in the mixture material), you can set the **Standard State Enthalpy** and **Reference Temperature** in the *Create/Edit Materials Dialog Box* (p. 1736).

8.11. Standard State Entropies

If you are using the finite-rate model with reversible reactions (see The Laminar Finite-Rate Model in the Theory Guide), you will need to define the standard state entropy, s_j^0 for each species j. These inputs are used to define the mixture entropy as

$$S = \sum_j m_j \left[s_j^0 + \int_{T_{ref,j}}^{T} \frac{c_{p,j}}{T} dT \right] \tag{8–74}$$

where $T_{ref,j}$ is the reference temperature at which s_j^0 is defined. Standard state entropies are input in units of J/kmol-K in SI units or in units of Btu/lb_mmol-$^\circ$R in British units.

For each species involved in the reaction (i.e., each fluid material contained in the mixture material), you can set the **Standard State Entropy** and **Reference Temperature** in the *Create/Edit Materials Dialog Box* (p. 1736).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.12. Molecular Heat Transfer Coefficient

If you are modeling premixed combustion (see *Modeling Premixed Combustion* (p. 893)), the fluid material in your domain should be assigned the properties of the unburned mixture, including the molecular heat transfer coefficient (α in Equation 9–9 in the Theory Guide), which is also referred to as the thermal diffusivity. α is defined as $k/\rho c_p$, and values at standard conditions can be found in combustion handbooks (e.g., [*42*] (p. 2211)). To determine values at non-standard conditions, you will need to use a third-party 1D combustion program with detailed chemistry. You can set the **Molecular Heat Transfer Coefficient** in the *Create/Edit Materials Dialog Box* (p. 1736).

8.13. Kinetic Theory Parameters

You may choose to define the following properties using kinetic theory when the ideal gas law is enabled:

- viscosity (for fluids)
- thermal conductivity (for fluids)
- specific heat capacity (for fluids)
- mass diffusion coefficients (for multi-species mixtures)

If you are using kinetic theory for a fluid's viscosity (*Equation 8–21* (p. 412)), you will need to input the kinetic theory parameters σ and ε/k_B for that fluid. These parameters and are referred to by ANSYS FLUENT as the "characteristic length" and the "energy parameter" respectively.

- When kinetic theory is applied to calculation of a fluid's thermal conductivity only, no inputs are required.
- To calculate specific heat of a fluid using kinetic theory (*Equation 8–55* (p. 435)), you need to enter the degrees of freedom for the fluid material.
- If you use kinetic theory to define a mixture material's mass diffusivity (*Equation 8–69* (p. 445)), you will need to input σ_i and $(\varepsilon/k_B)_i$ for each chemical species i.

For additional information, please see the following sections:

8.13.1. Inputs for Kinetic Theory

8.13.1. Inputs for Kinetic Theory

The procedure for using kinetic theory is as follows:

1. Select **kinetic-theory** as the property specification method for the **Viscosity**, **Thermal Conductivity**, or heat capacity **Cp** of a fluid material, or for the **Mass Diffusivity** of a mixture material.
2. If the material for which you have selected the kinetic theory method for one or more properties is a fluid material, you must set the kinetic theory parameters for each of the constituent species (fluid materials).

   The parameters to be set are as follows:

   - **L-J Characteristic Length**
   - **L-J Energy Parameter**
   - **Degrees of Freedom** (only required if kinetic theory is used for specific heat)

   See the beginning of this section to find out which parameters are required to calculate each property using kinetic theory.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Characteristic length is defined in units of Angstroms. The energy parameter is defined in units of absolute temperature. Degrees of freedom is a dimensionless input. All kinetic theory materials can be found in the literature (e.g., [*34*] (p. 2210)).

8.14. Operating Pressure

Specification of the operating pressure affects your calculation in different ways for different flow regimes. This section presents information about the operating pressure, its relevance for different cases, and how to set it correctly.

For additional information, please see the following sections:

8.14.1. The Effect of Numerical Roundoff on Pressure Calculation in Low-Mach-Number Flow
8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure
8.14.3. Setting the Operating Pressure

8.14.1. The Effect of Numerical Roundoff on Pressure Calculation in Low-Mach-Number Flow

In low-Mach-number compressible flow, the overall pressure drop is small compared to the absolute static pressure, and can be significantly affected by numerical roundoff. To understand why this is true, consider a compressible flow with $M 1$. The pressure changes, Δp, are related to the dynamic head, $\frac{1}{2}\gamma p M^2$, where p is the static pressure and γ is the ratio of specific heats. This gives the simple relationship $\Delta p/p \sim M^2$, so that $\Delta p/p \rightarrow 0$ as $M \rightarrow 0$. Therefore, unless adequate precaution is taken, low-Mach-number flow calculations are very susceptible to roundoff error.

8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure

ANSYS FLUENT avoids the problem of roundoff error (discussed in *The Effect of Numerical Roundoff on Pressure Calculation in Low-Mach-Number Flow* (p. 449)) by subtracting the operating pressure (generally a large pressure roughly equal to the average absolute pressure in the flow) from the absolute pressure, and using the result (termed the gauge pressure). The relationship between the operating pressure, gauge pressure, and absolute pressure is shown below. The absolute pressure is simply the sum of the operating pressure and the gauge pressure:

$$p_{abs} = p_{op} + p_{gauge} \tag{8–75}$$

All pressures that you specify and all pressures computed or reported by ANSYS FLUENT are gauge pressures.

8.14.3. Setting the Operating Pressure

8.14.3.1. The Significance of Operating Pressure

Operating pressure is significant for incompressible ideal gas flows because it directly determines the density: the incompressible ideal gas law computes density as $\rho = \frac{p_{op}}{\frac{R}{M_w}T}$. You must therefore be sure to set the operating pressure appropriately.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operating pressure is significant for low-Mach-number compressible flows because of its role in avoiding roundoff error problems, as described in *Operating Pressure, Gauge Pressure, and Absolute Pressure* (p. 449). Ensure to set the operating pressure appropriately. For time-dependent compressible flows, you may want to specify a floating operating pressure instead of a constant operating pressure. See *Floating Operating Pressure* (p. 507) for details.

Operating pressure is less significant for higher-Mach-number compressible flows. The pressure changes in such flows are much larger than those in low-Mach-number compressible flows, so there is no real problem with roundoff error and there is therefore no real need to use gauge pressure. In fact, it is common convention to use absolute pressures in such calculations. Since ANSYS FLUENT always uses gauge pressure, you can simply set the operating pressure to zero, making gauge and absolute pressures equivalent.

If the density is assumed constant or if it is derived from a profile function of temperature, the operating pressure is not used in the density calculation.

Note that the default operating pressure is 101325 Pa.

8.14.3.2. How to Set the Operating Pressure

The criteria for choosing a suitable operating pressure are based on the Mach-number regime of the flow and the relationship that is used to determine density. For example, if you use the ideal gas law in an incompressible flow calculation (e.g., for a natural convection problem), you should use a value representative of the mean flow pressure.

To place this discussion in perspective, *Table 8.1: Recommended Settings for Operating Pressure* (p. 450) shows the recommended approach for setting operating pressures. The default operating pressure is 101325 Pa.

Table 8.1 Recommended Settings for Operating Pressure

Density Relationship	Mach Number Regime	Operating Pressure
ideal gas law	$M > 0.1$	0 or ≈ mean flow pressure
ideal gas law	$M < 0.1$	≈ mean flow pressure
profile function of temperature	incompressible	not used
constant	incompressible	not used
incompressible ideal gas law	incompressible	≈ mean flow pressure

You will set the **Operating Pressure** in the *Operating Conditions Dialog Box* (p. 1806).

Cell Zone Conditions → **Operating Conditions...**

8.15. Reference Pressure Location

For incompressible flows that do not involve any pressure boundaries, ANSYS FLUENT adjusts the gauge pressure field after each iteration to keep it from floating. This is done using the pressure in the cell located at (or nearest to) the reference pressure location. The pressure value in this cell is subtracted from the entire gauge pressure field; as a result, the gauge pressure at the reference pressure location is always zero. If pressure boundaries are involved, the adjustment is not needed and the reference pressure location is ignored.

The reference pressure location is, by default, the cell center at or closest to (0,0,0). There may be cases in which you might want to move the reference pressure location, perhaps locating it at a point where the absolute static pressure is known (e.g., if you are planning to compare your results with experimental data).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change the location, enter new (**X,Y,Z**) coordinates for **Reference Pressure Location** in the *Operating Conditions Dialog Box* (p. 1806).

Cell Zone Conditions → **Operating Conditions...**

For additional information, please see the following sections:
8.15.1. Actual Reference Pressure Location

8.15.1. Actual Reference Pressure Location

For cases that do not have pressure-related boundary conditions (e.g., pressure inlet, pressure outlet, pressure far-field, etc.), you need to specify the **Reference Pressure Location** at a point in the problem domain. Internally, ANSYS FLUENT sets the location of the reference pressure at a slightly different nearby location. Thus, the actual location used as the pressure reference is different than that of your input value. To report the actual reference pressure location that ANSYS FLUENT uses, use the following text command:

`define` → `operating-conditions` → `used-ref-pressure-location`

Important

- This text command is available only when the case is initialized and has no pressure-related boundary zones.
- Reporting the actual reference pressure location is not available through the graphical user interface.

8.16. Real Gas Models

Some engineering problems involve fluids that do not behave as ideal gases. For example, at very high-pressure or very low-temperature conditions (e.g., the flow of a refrigerant through a compressor) the flow cannot typically be modeled accurately using the ideal-gas assumption. Therefore, the real gas model allows you to solve accurately for the fluid flow and heat transfer problems where the working fluid behavior deviates from the ideal-gas assumption.

ANSYS FLUENT provides three real gas options for solving these types of flows:

- *Cubic Equation of State Models* (p. 454)
- *The NIST Real Gas Models* (p. 468)
- *The User-Defined Real Gas Model* (p. 472)

All the models allow the user to solve for either a single-species fluid flow or a multiple-species mixture fluid flow.

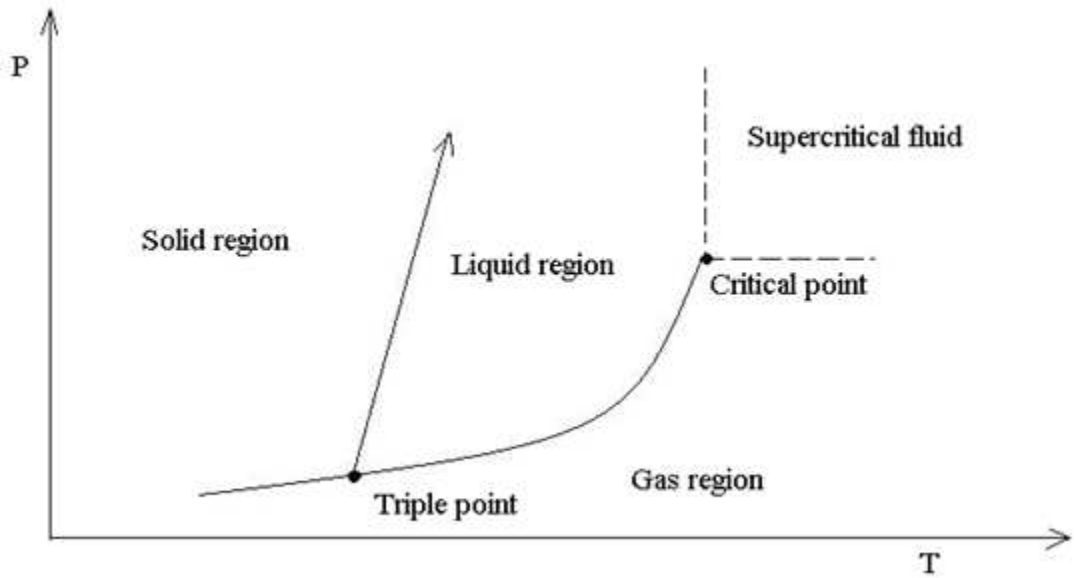
For additional information, please see the following sections:
8.16.1. Introduction
8.16.2. Choosing a Real Gas Model
8.16.3. Cubic Equation of State Models
8.16.4. The NIST Real Gas Models
8.16.5. The User-Defined Real Gas Model

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.1. Introduction

The states at which a pure material can exist can be graphically represented in diagrams of pressure vs. temperature (PT diagrams) and pressure vs. molecular or specific volume (PV diagrams). Homogeneous fluids are normally divided into two classes, liquids and gases. However the distinction cannot always be sharply drawn, because the two phases become indistinguishable at what is called the critical point. A typical pressure-temperature (PT) diagram of a pure material is shown in *Figure 8.30* (p. 452).

Figure 8.30 Typical PT Diagram of a Pure Material



This figure shows the single phase regions, as well as the conditions of P and T where two phases coexist. Thus the solid and the gas region are divided by the sublimation curve, the liquid and gas regions by the vaporization curve, and the solid and liquid regions by the fusion curve. The three curves meet at the triple point, where all three phases coexist in equilibrium. Although the fusion curve continues upward indefinitely, the vaporization curve terminates at the critical point. The coordinates of this point are called the critical pressure P_c and critical temperature T_c. These represent the highest temperature and pressure at which a pure material can exist in vapor-liquid equilibrium. At temperatures and pressures above the critical point, the physical property differences that differentiate the liquid phase from the gas phase become less defined. This reflects the fact that, at extremely high temperatures and pressures, the liquid and gaseous phases become indistinguishable. This new phase, which has some properties that are similar to a liquid and some properties that are similar to a gas, is called a supercritical fluid.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.31 Typical PV Diagram of a Pure Material

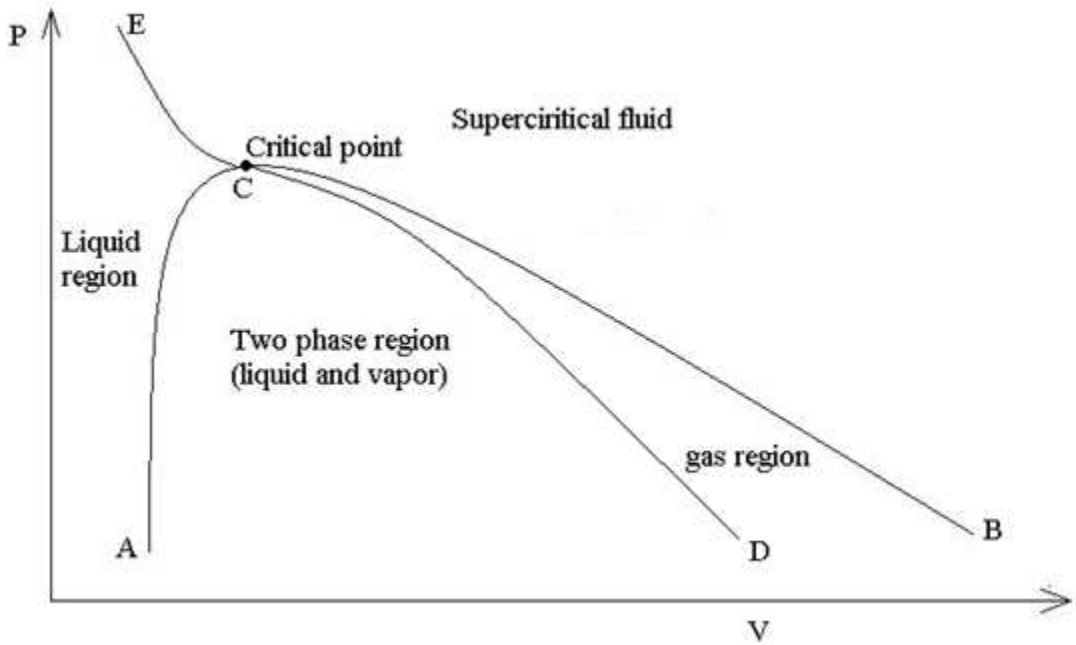


Figure 8.31 (p. 453) presents a typical diagram of pressure versus molar or specific volume (PV diagram) of a pure material. The dome shaped curve ACD is called the saturation dome and separates the single phase regions in the diagram; curve AC represents the saturated liquid and curve CD the saturated vapor. The area under the saturation dome ACD is the two-phase region and represents all possible mixtures of vapor and liquid in equilibrium. Curve ECB is the critical isotherm and exhibits a horizontal inflection at point C at the top of the dome. This is the critical point. The specific volume corresponding to the critical point, is called the critical specific volume V_c. The conditions to the right of the critical isotherm ECB correspond to super-critical fluid.

8.16.2. Choosing a Real Gas Model

The equation of state is the mathematical expression that relates pressure, molar or specific volume, and temperature for any pure homogeneous fluid in equilibrium states.

The simplest equation of state is the ideal gas law, which is approximately valid for the low pressure gas region of the PT and PV diagrams. Ideal gas behavior can be expected when

$$P/P_c \ll 1$$

or

$$T/T_c > 2 \text{ and } P/P_c < 1$$

If your flow conditions correspond to either of those cases, you may use the ideal gas law in your simulation.

Another idealization, that of the incompressible fluid, can be employed for the low pressure region of the liquid phase. A constant density option is the appropriate selection in that case.

However, both of these approaches are not good approximations for flow conditions close to and beyond the critical point, where the fluid behavior cannot be described by the ideal gas, or the incompressible liquid

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assumptions. We refer to a fluid under those conditions as a real fluid, or a real gas and more complex relations are used for the determination of its physical and thermodynamic properties.

ANSYS FLUENT provides the following options for solving real fluid problems:

- The cubic equation of state models can be used to solve problems in the gas and supercritical fluid regimes. The models are not available for the two-phase region under the phase dome. For further details see *Cubic Equation of State Models* (p. 454).
- The NIST real gas model can be used to solve problems in the liquid, or gas and supercritical fluid regimes. The model does not allow modeling of the two-phase region. For further details see *The NIST Real Gas Models* (p. 468).
- The User-Defined real gas model allows you to solve problems in all regimes, as long as appropriate relationships are provided through the User-Defined real gas functions. For further details see *The User-Defined Real Gas Model* (p. 472).

The concepts presented in this section for pure materials are also extended to multi-component mixtures with the introduction of appropriate composition dependent parameters in the real gas equations of state and the material property models. All the real-gas modeling options above allow for either single-species or multi-component flow modeling. In addition, you may solve reacting flow problems with the cubic equations of state models and the User-Defined real gas functions.

8.16.3. Cubic Equation of State Models

8.16.3.1. Overview and Limitations

An equation of state is a thermodynamic equation, which provides a mathematical relationship between two or more state functions associated with the matter, such as its temperature, pressure, volume, or internal energy. One of the simplest equations of state for this purpose is the ideal gas law, which is roughly accurate for gases at low pressures and high temperatures. However, this equation becomes increasingly inaccurate at higher pressures and lower temperatures, and fails to predict condensation from a gas to a liquid.

Introduced in 1949, the Redlich-Kwong equation of state [*67*] (p. 2212) was a considerable improvement over other equations of that time. It is an analytic cubic equation of state and is still of interest primarily due to its relatively simple form. The original form is

$$P=\frac{RT}{V-b}-\frac{\alpha_0}{V(V+b)T_r^{0.5}} \tag{8–76}$$

where

P = absolute pressure (Pa)

R = universal gas constant

V = specific molar volume ($\mathrm{m}^3/\mathrm{kmol}$)

T = temperature (K)

T_r = reduced temperature $\frac{T}{T_c}$, where T_c is the critical temperature

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

α_0 and b are constants related directly to the fluid critical pressure and temperature.

Many investigators have attempted to improve the accuracy of the Redlich-Kwong equation. ANSYS FLUENT has adopted the original form of the Redlich-Kwong equation, as well as the following modified forms:

- The Soave-Redlich-Kwong [*104*] (p. 2214) equation is a three-parameter equation of state, which can be applied for vapor, supercritical, and liquid property predictions. It has found wide acceptance mainly in the oil and gas industry and requires knowledge of the critical temperature, critical pressure, and acentric factor. It was developed by replacing the Redlich-Kwong attractive coefficient defined as $\alpha = \alpha_0 / T_r^{0.5}$ with a two-parameter form $\alpha(T_r, \omega)$, where ω is the acentric factor.
- The Peng-Robinson equation [*105*] (p. 2214) is a three-parameter equation of state, also requiring the critical temperature, critical pressure and acentric factor parameters. It is thought to perform as well as the Soave-Redlich-Kwong equation, with an advantage in the prediction of the liquid densities.
- The Aungier-Redlich-Kwong [*8*] (p. 2209) equation provides improved predictions for vapor and supercritical fluids near the critical point, as well as for materials with a negative value of the acentric factor. The Aungier modified form is a four parameter equation and requires the critical specific volume in addition to the critical temperature, critical pressure and acentric factor.

The following limitations exist in ANSYS FLUENT for all cubic equation of state models:

- Pressure-inlets, mass flow-inlets, and pressure-outlets are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions should not be used with the real gas models.
- The cubic equation of state real gas models are compatible with the Eulerian multiphase models.

Note

Please note the following restrictions:

– Only one phase can be a real gas.

– Only **constant-rate** and **user-defined Mass Transfer Mechanisms** are available.

- The cubic equation of state models are compatible with the Lagrangian Dispersed Phase Models. If you are modeling droplet or multicomponent particles, please note that the current formulation does not take into account the near-critical point phenomena, which means that accurate results can be obtained for droplet temperatures below $T_{\lim} = 0.9 T_c$. See *Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models* (p. 465) for more details.
- The real gas models cannot be used with the premixed, partially premixed, inert, and composition PDF transport combustion models.
- The cubic equation of state real gas models can be used with the following species models:
  - Species transport. Chemical reactions can be modeled with the finite rate and eddy dissipation models. Please note that the **Dimension Reduction** model is not available with the real gas models.
  - Non-premixed model. Please note that the **Compressibility Effects** option must be enabled in the **Species Model** dialog box in order for the real-gas models to be available. In addition the following restrictions apply when the non-premixed model is used together with a cubic equation of state real-gas model:
    - → The empirical stream options cannot be used, as for the empirical species the critical properties cannot be defined.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Condensed species such as h2o<l> and c<s> are not supported and should be excluded from the PDF table, so please take care to add all condensed species for your system in the excluded species list prior to the PDF table generation.

8.16.3.2. Equation of State

The general form of pressure P for the cubic equation of state models is written as [*8*] (p. 2209) :

$$P=\frac{RT}{V-b+c}-\frac{\alpha}{(V^2+\delta V+\varepsilon)} \quad \textbf{(8–77)}$$

where

P = absolute pressure (Pa)

V = specific molar volume ($\mathrm{m}^3/\mathrm{kmol}$)

T = temperature (K)

R = universal gas constant

The coefficients α, b, c, δ, and ε are given for each equation of state as functions of the critical temperature T_c, critical pressure P_c, acentric factor ω and critical specific volume V_c. Note that the attractive coefficient α also has a temperature dependence, which varies for each equation of state model, and is commonly written as $\alpha=\alpha(T)$.

Redlich-Kwong Equation [*106*] (p. 2214):

$$\alpha(T)=\frac{\alpha_0}{(T/T_c)^{0.5}} \quad \textbf{(8–78)}$$

$$\alpha_0=\frac{0.42747R^2T_c^2}{P_c} \quad \textbf{(8–79)}$$

$$b=\frac{0.08664RT_c}{P_c} \quad \textbf{(8–80)}$$

The parameter δ is set equal to b, while c and ε are set to 0. The Redlich-Kwong equation is the simplest of the cubic equations of state in ANSYS FLUENT and requires two parameters only, the critical temperature T_c and the critical pressure P_c.

Soave-Redlich-Kwong Equation[*104*] (p. 2214):

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\alpha(T)=\alpha_0[1+n(1-(T/T_c)^{0.5})]^2 \quad \textbf{(8–81)}$$

$$n=0.48+1.574\omega-0.176\omega^2 \quad \textbf{(8–82)}$$

α_0 and b are given by *Equation 8–79* (p. 456) and *Equation 8–80* (p. 456) respectively. As in the original Redlich-Kwong equation the parameter δ is set equal to b, while c and ε are set to 0. The Soave-Redlich-Kwong requires three parameters, the critical temperature T_c, the critical pressure P_c, and the acentric factor ω.

Peng-Robinson Equation [*105*] (p. 2214):

$$\alpha_0=\frac{0.457247R^2T_c^2}{P_c} \quad \textbf{(8–83)}$$

$$b=\frac{0.07780RT_c}{P_c} \quad \textbf{(8–84)}$$

The function $\alpha(T)$ is given by *Equation 8–81* (p. 457), with n provided in *Equation 8–85* (p. 457) as follows:

$$n=0.37464+1.54226\omega-0.26992\omega^2 \quad \textbf{(8–85)}$$

In the Peng-Robinson equation δ is set equal to $2b$, ε is equal to $-b^2$, and c is set to 0.

Similar to the Soave-Redlich-Kwong equation, the Peng-Robinson equation is a three-parameter equation and requires the critical temperature T_c, the critical pressure P_c, and the acentric factor ω.

Aungier-Redlich-Kwong Equation [*8*] (p. 2209):

$$\alpha(T)=\alpha_0\left(\frac{T}{T_c}\right)^{-n} \quad \textbf{(8–86)}$$

$$n=0.4986+1.1735\omega+0.4754\omega^2 \quad \textbf{(8–87)}$$

α_0 and b are given by *Equation 8–79* (p. 456) and *Equation 8–80* (p. 456), respectively. As in the original Redlich-Kwong equation, the parameter δ is set equal to b and ε is set to 0. Parameter c is given by:

$$c=\frac{RT_c}{P_c+\frac{\alpha_0}{V_c(V_c+b)}}+b-V_c \quad \textbf{(8–88)}$$

The Aungier-Redlich-Kwong equation requires four parameters, namely the critical temperature T_c, the critical pressure P_c, the critical specific volume V_c, and the acentric factor ω.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations

Enthalpy, entropy, and specific heat are computed in terms of the relevant ideal gas properties and the departure functions. The departure function F_{dep} of any conceptual property F is defined as [67] (p. 2212)

$$F_{dep}=F_{ideal}(T,P)-F(T,P) \quad \textbf{(8–89)}$$

where F_{ideal} is the value of the property as computed from the ideal gas relations. The departure function F_{dep} can be derived from basic thermodynamic relations and the equation of state.

Following the above definition, the enthalpy H for the equations of state models is given by the following equations [8] (p. 2209):

$$H=H_{ideal}-\frac{H_{dep}}{MW} \quad \textbf{(8–90)}$$

$$H_{dep}=-PV+RT-\frac{T\frac{\partial a}{\partial T}-a}{\Delta}ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \quad \textbf{(8–91)}$$

$$\Delta=(\delta^2-4\epsilon)^{0.5} \quad \textbf{(8–92)}$$

where,

H_{ideal} = ideal gas enthalpy at temperature T (J/kg)

H_{dep} = departure enthalpy (J/kmol)

MW = mean molecular weight (Kg/kmol)

P = pressure (Pa)

V = specific molar volume (m^3/kmol)

R = universal gas constant

α, δ, and ε are computed using *Equation 8–78* (p. 456) – *Equation 8–88* (p. 457) depending on the equation of state model.

Similarly, the departure internal energy U_{dep} can be shown to be

$$U_{dep}=-\frac{T\frac{\partial a}{\partial T}-a}{\Delta}ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \quad \textbf{(8–93)}$$

where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

U_{dep} = departure internal energy (J/kmol)

T = temperature (K)

V = specific molar volume (m^3/kmol)

Δ is given by *Equation 8–92* (p. 458)and α, δ, and ε are computed using *Equation 8–78* (p. 456) – *Equation 8–88* (p. 457) depending on the equation of state model.

The specific heat c_p can be computed from the ideal specific heat $c_{p,ideal}$ and the departure specific heat at constant volume $c_{v,dep}$ as follows:

$$c_p = c_{p,ideal} - \frac{c_{p,dep}}{MW} \quad \textbf{(8–94)}$$

$$c_{p,dep} = c_{v,dep} - R - T\frac{(\partial V/\partial T)^2}{\partial V/\partial P} \quad \textbf{(8–95)}$$

where

MW = mean molecular weight (kg/kmol)

P = pressure (Pa)

T = temperature (K)

V = specific molar volume (m^3/kmol)

R = universal gas constant

In *Equation 8–95* (p. 459) $c_{v,dep}$ is computed by differentiating the equation of departure internal energy (*Equation 8–93* (p. 458)) with respect to T, and the partial derivatives of the specific volume are computed by differentiating *Equation 8–77* (p. 456) appropriately.

The entropy S is computed by

$$S = S_{ideal,0} - \frac{S_{dep}}{MW} \quad \textbf{(8–96)}$$

$$S_{dep} = -R\ln\left(\frac{V-b+c}{V_0}\right) - \frac{\frac{da}{dT}}{\Delta}\ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \quad \textbf{(8–97)}$$

where

$S_{ideal,0}$ = ideal gas entropy at temperature T and reference pressure (J/kg/K)

V_0 = ideal gas specific molar volume at temperature T and the reference pressure (m^3/kmol)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P = pressure (Pa)

T = temperature (K)

V = specific molar volume (m^3/kmol)

MW = mean molecular weight (kg/kmol)

Δ is given by *Equation 8–92* (p. 458) and α, δ, and ε are computed using *Equation 8–78* (p. 456) – *Equation 8–88* (p. 457), depending on the equation of state model. Note that the pressure term in *Equation 8–96* (p. 459) cancels out as both $S_{ideal,0}$ and V_0 are evaluated at the reference pressure.

8.16.3.4. Critical Constants for Pure Components

Equations describing real-gas properties require the knowledge of the critical constants for pure components and mixtures. These comprise the critical temperature (T_c), critical pressure (P_c), critical specific volume (V_c), and the acentric factor (ω).

Several critical constants for fluid materials in the ANSYS FLUENT property database `propdb.scm` have been compiled from a variety of sources available in the open literature [*67*] (p. 2212), [*59*] (p. 2212), [*60*] (p. 2212), [*81*] (p. 2213), [*82*] (p. 2213), [*85*] (p. 2213), [*7*] (p. 2209).

For those fluid materials, for which the critical properties have not been found in the open literature, these have been estimated using the commercially available software CRANIUM by Molecular Knowledge Systems Inc. [*4*] (p. 2209) : `http://www.molknow.com/Cranium/cranium.htm`

Critical property values for many hydrocarbon and nitrogenous radical species have been obtained from Tang and Brezinsky [*91*] (p. 2213). Where the critical properties for the radicals were not available in the literature, these were estimated using a modification of the Joback method [*67*] (p. 2212). This assumes that the radical site constitutes a distinct group with zero group contribution and utilizes the group contribution values for stable species.

The critical properties of coal volatiles have been estimated assuming that the volatiles can be approximated by a mixture of CO, CO_2, H_2, CH_4, and C_2H_6 in such a way, that the atom composition and the net calorific value of the volatiles is similar to that of the assumed mixture. The critical properties of the lignite and biomass volatiles have been assumed equal to those of formaldehyde. The critical properties of diesel and jet-a fuels have been set equal to those of decane. The critical properties of kerosene have been set equal to those of dodecane.

8.16.3.5. Calculations for Mixtures

For the computation of properties in real-gas mixtures, ANSYS FLUENT follows the so called pseudocritical method. According to this method, the behavior and properties of a real gas mixture will be the same as that of a pure component, to which appropriate critical constants are assigned. These mixture critical constants are functions of the mixture composition and pure component critical properties, and are sometimes called pseudocritical constants, because their values are generally expected to be different from the true mixture critical constants that may be determined experimentally. However for computational purposes they are the appropriate critical constant values for the mixture. According to the pseudocritical method, ANSYS FLUENT applies *Equation 8–77* (p. 456) – *Equation 8–97* (p. 459) also for mixtures, where the critical temperature T_c, critical pressure P_c, critical specific volume V_c, and acentric factor ω, are replaced by the corresponding mixture critical constants, critical temperature T_{cm}, critical pressure P_{cm}, critical specific volume V_{cm}, and acentric factor ω_m.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following options are available in ANSYS FLUENT for the calculation of the mixture pseudocritical constants:

- The simplest rule for computing the pseudocritical constants C_{cm} for a real gas mixture is the mole fraction average [67] (p. 2212). This method is recommended for mixtures where the pure component critical properties for all components are not very different:

$$C_{cm} = \sum_{i=1}^{k} x_i C_{ci} \tag{8–98}$$

  where

  C_{cm} = mixture pseudocritical constant (temperature, pressure, specific volume or acentric factor)

  C_{ci} = critical constant of component i (temperature, pressure, specific volume or acentric factor)

  x_i = mole fraction of component i

  k = number of components in mixture

- An alternative approach is based on the one-fluid van der Waals mixing rules as expressed in [106] (p. 2214). According to this approach, in order to apply the equation of state models to mixtures, the coefficients a and b in *Equation 8–77* (p. 456) are replaced by composition dependent expressions as follows:

$$(\alpha_m)^{0.5} = \sum_i x_i (\alpha_i)^{0.5} \tag{8–99}$$

$$b_m = \sum_i x_i b_i \tag{8–100}$$

  where

  x_i = mole fraction of species i

  With the appropriate expressions from *Equation 8–78* (p. 456) – *Equation 8–88* (p. 457) for each equation of state, and assuming a mixture acentric factor ω_m for the evaluation of parameter n in *Equation 8–82* (p. 457), *Equation 8–85* (p. 457), and *Equation 8–87* (p. 457), the mixing rules *Equation 8–99* (p. 461) and *Equation 8–100* (p. 461) can be rearranged to yield direct expressions of the mixture critical properties as functions of the mole fractions and the component critical properties [8] (p. 2209).

  The resulting expressions for the mixture critical specific temperature T_{cm} are as follows:

  – Soave-Redlich-Kwong and Peng-Robinson models

$$T_{cm} = \frac{\left[\sum_i \left(x_i \frac{T_{ci}}{P_{ci}^{0.5}}\right)\right]^2}{\sum_i \left(\frac{x_i T_{ci}}{P_{ci}}\right)} \tag{8–101}$$

  – Redlich-Kwong model and Aungier-Redlich-Kwong model:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T_{cm}^{1+n} = \frac{\left[\sum_i \left(x_i \sqrt{\frac{T_{ci}^{(2+n)}}{P_{ci}}} \right)\right]^2}{\sum_i \left(\frac{x_i T_{ci}}{P_{ci}} \right)} \tag{8–102}$$

where for the Aungier-Redlich-Kwong model n is obtained from *Equation 8–87* (p. 457) as function of the mixture acentric factor ω_m. For the Redlich-Kwong model $n = 0.5$.

The mixture critical specific pressure P_{cm} is computed as:

$$P_{cm} = \frac{T_{cm}}{\sum_i \left(\frac{x_i T_{ci}}{P_{ci}} \right)} \tag{8–103}$$

The mixture critical specific volume V_{cm} is computed as:

$$V_{cm} = \sum_I \frac{x_i P_{ci} V_{ci}}{T_{ci}} \left(\frac{T_{cm}}{P_{cm}} \right) \tag{8–104}$$

The notation for *Equation 8–101* (p. 461) to *Equation 8–104* (p. 462) is

T_{cm} = mixture pseudocritical temperature (K)

P_{cm} = mixture pseudocritical pressure (Pa)

V_{cm} = mixture pseudocritical molar volume (m^3/kmol

T_{ci} = critical temperature for component i (K)

P_{ci} = critical pressure for component i (Pa)

V_{ci} = critical molar volume for component i (m^3/kmol)

x_i = mole fraction for component i

8.16.3.5.1. Using the Cubic Equation of State Real Gas Models

For single or multi-component flows, you will activate the cubic equation of state real gas models by selecting **real-gas-soave-redlich-kwong**, **real-gas-peng-robinson**, **real-gas-aungier-redlich-kwong**, or **real-gas-redlich-kwong** from the **Density** drop-down list in the **Create/Edit Materials** dialog box.

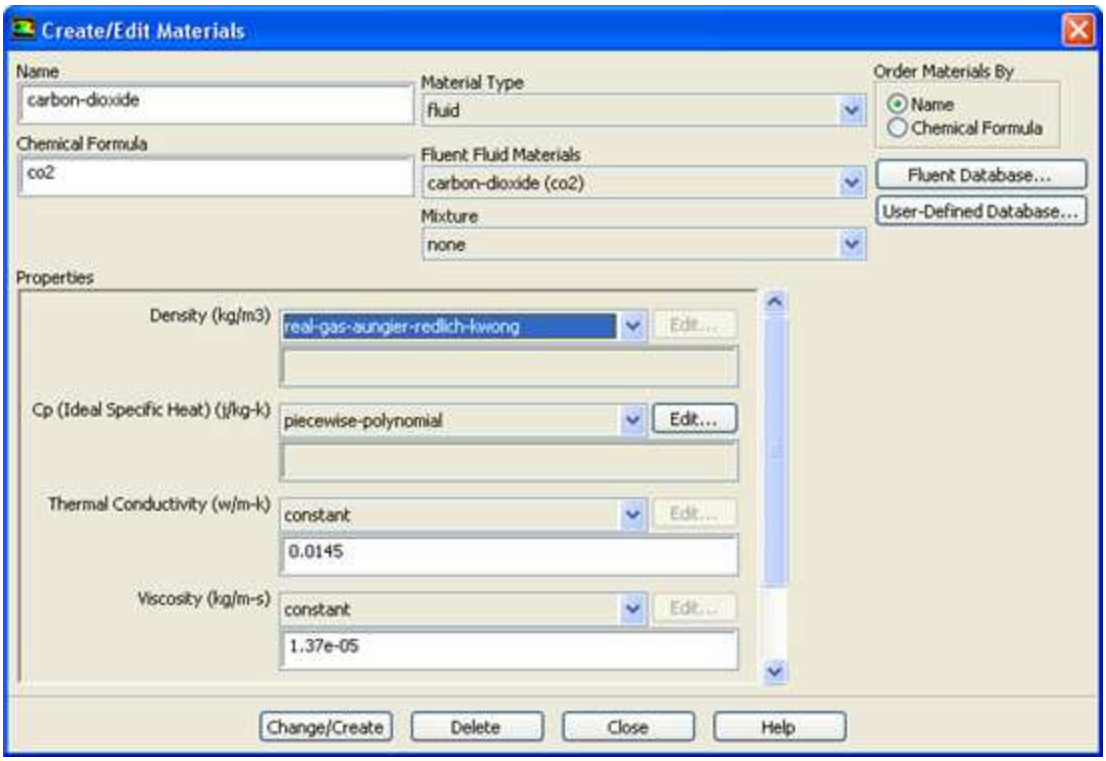
Materials → **Create/Edit...**

The required inputs for the cubic equation of state real gas models for single component flow and mixtures are described below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Single Component Flow

Figure 8.32 The Cubic Equation of State Model for a Real-Gas Fluid



When any of the cubic equation of state models is enabled, enter the following material properties in the dialog box:

- ideal specific heat
- molecular weight
- standard state entropy
- reference temperature
- critical temperature
- critical pressure
- critical specific volume
- acentric factor

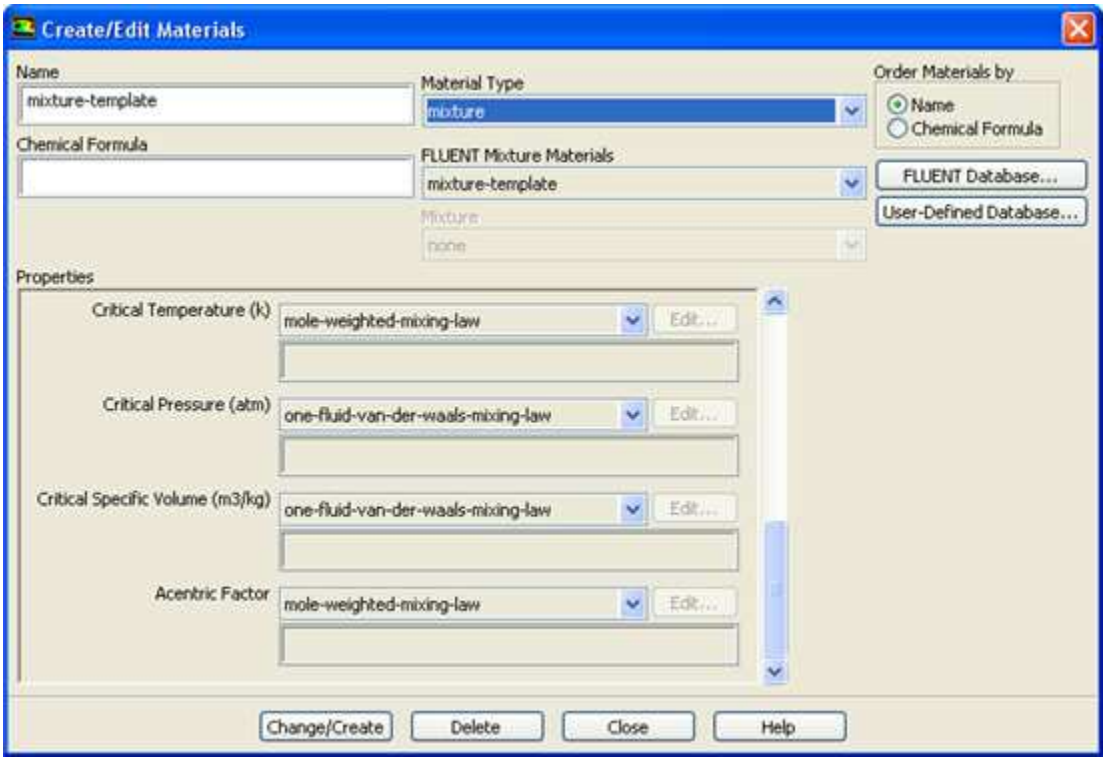
Important

Your inputs for the specific heat in the **Materials** dialog box will now be used to compute the ideal property functions H_{ideal}, $c_{p,ideal}$, and $S_{ideal,0}$ in *Equation 8–90* (p. 458), *Equation 8–94* (p. 459), and *Equation 8–96* (p. 459), respectively. In ANSYS FLUENT the departure properties will be computed and added to the ideal part, to yield the real gas specific heat, enthalpy, and entropy.

Mixtures

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.33 The Cubic Equation of State Model for a Real-Gas Mixture



When one of the cubic equation of state models is selected from the **Density** drop-down list, specify the following material properties for the mixture material in the dialog box:

- ideal specific heat
- critical temperature
- critical pressure
- critical specific volume
- acentric factor

You also need to enter the following material properties for each of the mixture components in the dialog box:

- molecular weight
- standard state entropy
- reference temperature

When you are modeling a real-gas mixture, the following methods are available for the mixture critical constants:

- **constant**: defines a constant critical temperature, critical pressure, critical specific volume, or acentric factor for the mixture material.
- **mole-weighted-mixing-law**: applies *Equation 8–98* (p. 461) for critical temperature, critical pressure, critical specific volume, or acentric factor for the mixture material.
- **one-fluid-van-der-waals-mixing-law** applies *Equation 8–101* (p. 461) and *Equation 8–102* (p. 462) for the mixture critical temperature (depending on the real-gas model), *Equation 8–103* (p. 462) for the mixture critical pressure, and *Equation 8–104* (p. 462) for the mixture critical density.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- Ensure to click the **Change/Create** button so that all the above mentioned properties are uploaded and are visible in the interface.
- If you have selected **mixing-law** for the mixture ideal specific heat you will also need to enter the ideal specific heat values for the individual mixture components. If you have not selected **constant** as the option for any of the critical properties, you will need to enter the corresponding pure component critical properties for the mixture components.

8.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models

The flow modeling of real-gas flow is more complex and challenging than simple ideal-gas flow. Therefore, the solution might converge at a slower rate with real-gas flow than when running ideal-gas flow. It is recommended that you first attempt to converge your solution using first-order discretization then switch to second-order discretizations and re-iterate to convergence.

It is important to realize that the cubic equations of state real gas model is not available for two-phase flows where liquid and vapor coexist. Thus the flow conditions you are prescribing must fall within the range of the model. In case the flow conditions in your case fall inside the saturation dome, the properties are computed for the vapor phase up to the vapor spinodal curve, which defines the boundary beyond which the equation of state is no longer valid, because the local derivative of pressure with respect to volume becomes positive. State points predicted inside the dome, up to the spinodal curve, are called “metastable” because normally they only temporarily exist in small local regions until phase change occurs. For cases without phase change these states can occur and continue to persist depending on the problem setup. In some instances, the actual converged state is just within the superheated vapor limits but only transitory inside the saturation dome.

While the cubic equations of state real gas models are not available for two-phase flows, they can be applied to conditions of supercritical pressure $P > P_c$ and subcritical temperature $T < T_c$, where the fluid is in the liquid state. The supercritical liquid can co-exist with gas and supercritical fluid in the same simulation. In those cases, phase change may take place, without any of the flow conditions falling inside the saturation dome. For multicomponent simulations, when **Diffusion Energy Source** is enabled in the **Species Model** dialog box, the species energy diffusion is by default suppressed in the supercritical liquid regime and across the gas-liquid boundary. The diffusion energy source can be included in the liquid regime using the `define/models/species/liquid-energy-diffusion`? text command.

Finally, when you initialize the flow, ensure that the flow conditions fall within the vapor or supercritical flow conditions that are supported by the cubic equations of state model.

8.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models

If your simulation contains a liquid stream, the appropriate modeling approach in the various operating condition regimes is as follows:

1. For $T > T_c$ a liquid phase does not exist.
2. In the region $P > P_c$, you should define the flow streams directly in the boundary conditions by setting the appropriate pressure and temperature and the properties will be computed directly by the cubic EOS. This is applicable for both $T < T_c$ (supercritical liquid) and $T > T_c$ (supercritical fluid) regimes.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. For $P < P_c$ a stream can exist in liquid phase when its temperature $T < T_c$. If you would like to model phase change in this regime, the following recommendations apply:
   - The DPM droplet and multi-component models are adequate and recommended for the conditions away from the critical point. This regime can be defined as $T < 0.9T_c$, where the liquid physical properties can be assumed independent of pressure.
   - The region $T_c > T > 0.9T_c$ is characterized by near-critical-point phenomena, such as strong liquid density and specific heat dependence on both temperature and pressure. The DPM model can also be used in this regime, but you should be cautious, as it will not take into consideration the near-critical-point behavior. In addition, the applicability of the evaporation and boiling rate equations is questionable in this regime.

The droplet saturation vapor pressure corresponding to $T_{\lim} = 0.9T_c$ gives an indication of the maximum operating pressure for applicability of the DPM models for each droplet material. These pressure limits are listed in *Table 8.2: Pressure Limits for Droplet Materials in ANSYS FLUENT's Database* `prodb.scm` (p. 466) for many of the droplet materials in ANSYS FLUENT's `propdb.scm` materials database.

Table 8.2 Pressure Limits for Droplet Materials in ANSYS FLUENT's Database `prodb.scm`

Material	Tc (K)	Pc (MPa)	Normal BP (K)	Tlim (K)	Pressure Limit (MPa)
Argon	150.86	4.89	87.30	135.77	2.66
Benzene	562.00	4.89	353.00	505.80	2.35
Helium	5.30	0.23	4.20	4.77	0.16
Hydrogen	32.98	1.30	20.40	29.68	0.68
Methyl-alcohol	512.00	8.10	338.00	460.80	3.18
Heptanes	540.00	2.74	371.00	486.80	1.22
Hexane	507.00	3.02	342.00	456.30	1.36
Octane	569.00	2.49	399.00	512.10	1.08
Pentane	470.00	3.37	309.00	423.00	1.59
Nitrogen	126.00	3.40	77.40	113.40	1.73
Oxygen	154.00	5.04	90.20	138.60	2.62
Toluene	592.00	4.11	384.00	532.80	1.89
Water	647.00	22.00	373.00	582.30	9.71

For high pressure simulations the boiling point will be different from the normal boiling point, and for varying pressure applications the boiling point will vary with the droplet location in the domain.

When a cubic equation of state real gas model is enabled in a simulation which includes evaporating droplet particles, the boiling point is calculated from the vapor pressure data directly, as the temperature where the saturation vapor pressure equals the domain pressure. In addition, the latent heat of the evaporating or boiling droplet will vary with the droplet temperature.

The latent heat at temperature T_p is given by

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_{lat\_T} = -\int_{T_p}^{T_{bp}} c_{p,g} dT + H_{lat} + \int_{T_p}^{T_{bp}} c_{p,p} dT \quad \textbf{(8–105)}$$

where T_{bp} is the normal boiling point and H_{lat} is the latent heat at the normal boiling point.

In the **Create/Edit Materials** dialog box you will need to enter the **Normal Boiling Point (NBP)** and the **Latent Heat at NBP** for the **droplet-particle Material Type**. These inputs will be used for calculating the **Latent Heat** at the reference temperature (see Equation 16–302) and the latent heat in the droplet energy balance during vaporization and boiling according to *Equation 8–105* (p. 467).

Important

The constant property option is disabled for the **Saturation Vapor Pressure** property when a real gas model is used in the simulation. Also, ensure to enter the appropriate droplet saturation vapor pressure data to cover the complete pressure/temperature range in your model.

Finally when a cubic equation of state real-gas model is enabled in a simulation with droplet models, the condition for switching from the vaporization to the boiling law will be

$$P_{sat} > P \quad \textbf{(8–106)}$$

where P_{sat} is the saturation vapor pressure and P is the domain pressure. If $P_{sat} < P$ while in the boiling law, the model will switch back to the vaporization law.

8.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model

All postprocessing functions properly report and display the current thermodynamic and transport properties of the selected real gas model. The thermodynamic and transport properties controlled by the cubic equations of state real gas model include the following:

- density
- enthalpy
- entropy
- sound speed
- specific heat
- any quantities that are derived from the properties listed above (e.g., total quantities, ratio of specific heats)

In addition to the properties listed above, you can also report

- compressibility factor
- reduced temperature
- reduced pressure
- spinodal temperature

If you are modeling a real-gas mixture you can report the composition dependent mixture critical properties

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- critical temperature
- critical pressure
- critical specific volume
- acentric factor

8.16.4. The NIST Real Gas Models

The NIST real gas models use the National Institute of Standards and Technology (NIST) Thermodynamic and Transport Properties of Refrigerants and Refrigerant Mixtures Database Version 7.0 (REFPROP v7.0) to evaluate thermodynamic and transport properties of approximately 39 pure fluids or a mixture of these fluids.

The REFPROP v7.0 database is a shared library that is dynamically loaded into the solver when you activate one of the NIST real gas models in an ANSYS FLUENT session. Once the NIST real gas model is activated, control of relevant property evaluations is relinquished to the REFPROP database. All postprocessing functions will properly report and display the current thermodynamic and transport properties of the real gas.

8.16.4.1. Limitations of the NIST Real Gas Models

The following limitations exist for the NIST real gas model:

- Access to the *Create/Edit Materials Dialog Box* (p. 1736) from the navigation pane or **Define** menu is restricted. Therefore, if solid properties have to be set and modified, then it should be done in the *Create/Edit Materials Dialog Box* (p. 1736) before activating the real gas model.
- The NIST real gas model assumes that the fluid you will be using in your ANSYS FLUENT computation is superheated vapor, supercritical fluid, or liquid. Note that subcritical flow conditions, where vapor coexists with liquid in two-phase flow, are not supported. In addition, all fluid zones must contain the real gas; you cannot include a real gas and another fluid in the same problem.
- Pressure-inlet, mass flow-inlet, and pressure-outlet are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions should not be used with the real gas models.
- The mixture flow does not permit chemical reactions with the NIST real gas model.
- The real gas models cannot be used with any of the multiphase models. The model is compatible with the Lagrangian Dispersed Phase Models only for the massless and inert particle types. Please note that particle material properties have to be set in the *Create/Edit Materials Dialog Box* (p. 1736) before activating the real gas model.
- You cannot modify material properties in the REFPROP database libraries, or add custom materials to the NIST real gas model.

8.16.4.2. The REFPROP v7.0 Database

The NIST real gas model supports 83 pure fluids from the REFPROP database. These include 39 materials that were made available in the NIST web site later. The pure-fluid refrigerants and hydrocarbons that are supported by REFPROP v7.0 and used in the NIST real gas model are listed in *Table 8.3: Hydrocarbons and Refrigerants Supported by REFPROP v7.0* (p. 469) (the corresponding property file name appears in parentheses, where it does not coincide with the fluid name).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The database does not include transport property models for the following species: acetone, benzene, c4f10, c5fl2, cos, cyclohexane, cyclopropane, deuterium, fluorine, neopentane, nf3, propyne, r21, sf6, so2. As a result the NIST real gas model with those species can only be used for modeling inviscid flow.

The REFPROP v7.0 database employs accurate pure-fluid equations of state that are available from NIST. These equations are based on three models:

- modified Benedict-Webb-Rubin (MBWR) equation of state
- Helmholtz-energy equation of state
- extended corresponding states (ECS)

For a fluid that consists of a multispecies-mixture the thermodynamic properties are computed by employing mixing-rules applied to the Helmholtz energy of the mixture components.

Table 8.3 Hydrocarbons and Refrigerants Supported by REFPROP v7.0

1butene	acetone	ammonia	argon	benzene	butane
dodecane (c12.fld)	cis-butene (c2butene.fld)	c4f10	c5fl2	co	co2
cos	cyclohexane (cyclo-hex.fld)	cyclopropane (cyclopro.fld)	deuterium (d2.fld)	heavy water (d2o.fld)	decane
dimethylether (dme.fld)	ethane	ethanol	ethylene	fluorine	h2s
helium	heptane	hexane	hydrogen	ibutene	ihexane
ipentane	isobutene	krypton	methane	methanol	n2o
neon	neopentane (neopentn.fld)	nf3	nitrogen	nonane	octane
oxygen	parahydrogen (parahyd.fld)	pentane	propane	propylene (pro-pylen.fld)	propyne
r11	r113	r114	r115	r116	r12
r123	r124	r125	r13	r134a	r14
r141b	r142b	r143a	r152a	r21	r218
r22	r227ea	r23	r236ea	r236fa	r245ca
r245fa	r32	r365mfc	r41	rc318	sf6
so2	trans-butene (t2butene.fld)	toluene	water	xenon	

8.16.4.3. Using the NIST Real Gas Models

When you enable one of the NIST real gas models (single-species fluid or multiple-species mixture) and select a valid material, ANSYS FLUENT's functionality remains the same as when you model fluid flow and heat transfer using an ideal gas, with the exception of the *Create/Edit Materials Dialog Box* (p. 1736) (see below).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.4.3.1. Activating the NIST Real Gas Model

Activating one of the NIST real gas models is a two-step process. First you enable either the single-species NIST real gas model or the multi-species NIST real gas model, and then you select the fluid material from the REFPROP database.

1. Enabling the appropriate NIST real gas model:

   If you are solving for a single-species flow then you should enable the single-species NIST real gas model by typing the following text command at the ANSYS FLUENT console prompt:

   ```
   > define/user-defined/real-gas-models/nist-real-gas-model
   use NIST real gas? [no] yes
   ```

   If you are solving for multi-species mixture then you should enable the multi-species NIST real gas model by typing the following text command at the ANSYS FLUENT console prompt:

   ```
   > define/user-defined/real-gas-models/nist-multispecies-real-gas-model
   use multispecies NIST real gas? [no] yes
   ```

   The list of available pure-fluid materials you can select from will be displayed:

   ```
   1butene.fld     acetone.fld    ammonia.fld     argon.fld       benzene.fld
   butene.fld      c12.fld         c2butene.fld    c4fl0.fld       c5fl2.fld
   co.fld           co2.fld         cos.fld          cyclohex.fld    cyclopro.fld
   d2.fld           d2o.fld         decane.fld      dme.fld          ethane.fld
   ethanol.fld     ethylene.fld   fluorine.fld    h2s.fld          helium.fld
   heptane.fld     hexane.fld     hydrogen.fld    ibutene.fld     ihexane.fld
   ipentane.fld    isobutan.fld   krypton.fld     methane.fld     methanol.fld
   n2o.fld          neon.fld        neopentn.fld    nf3.fld          nitrogen.fld
   nonane.fld      octane.fld     oxygen.fld      parahyd.fld     pentane.fld
   propane.fld     propylen.fld   propyne.fld     r11.fld          r113.fld
   r114.fld         r115.fld        r116.fld         r12.fld          r123.fld
   r124.fld         r125.fld        r13.fld          r134a.fld        r14.fld
   r141b.fld        r142b.fld       r143a.fld        r152a.fld        r218.fld
   r21.fld          r22.fld         r227ea.fld       r23.fld          r236ea.fld
   r236fa.fld      r245ca.fld     r245fa.fld      r32.fld          r365mfc.fld
   r41.fld          rc318.fld       sf6.fld          so2.fld          t2butene.fld
   toluene.fld     water.fld      xenon.fld
   ```

2. Select material from the REFPROP database list:

   If the single-species real gas model is selected, then you need to enter the name of one fluid material when prompted:

   ```
   select real-gas data file [""] "r125.fld"
   ```

   Important

   You *must* enter the complete name of the material (including the `.fld` suffix) contained within quotes (`" "`).

   If the multiple-species real gas model is selected, then you need to enter the number of species in the mixture:

   ```
   Number of species [] 3
   ```

   followed by the name of each fluid selected from the list shown above:

   ```
   select real-gas data file [""] "nitrogen.fld"
   select real-gas data file [""] "co2.fld"
   select real-gas data file [""] "r22.fld"
   ```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Upon selection of a valid material (e.g., `r125.fld`), ANSYS FLUENT will load data for that material from a library of pure fluids supported by the REFPROP database, and report that it is opening the shared library (`librealgas.so`) where the compiled REFPROP database source code is located.

```
/usr/local/Fluent.Inc/fluent6.2/realgas/lib/r125.fld

Opening "/usr/local/Fluent.Inc/fluent6.2/realgas/
ultra/librealgas.so"...
Setting material "air" to a real-gas...

Matl name: "R125"
   : "pentafluoroethane !full name"
   : "354-33-6"
Mol Wt   : 120.021

Critical properties:
Temperature : 339.173 (K)
Pressure : 3.6177e+06 (Pa)
Density  : 4.779 (mol/L) 573.582 (kg/m^3)

Equation Of State (EOS) used:
Helmholtz Free Energy (FEQ)
EOS:"FEQ Helmholtz equation of state for R-125 of Lemmon and Jacobsen (2002)."

EOS Range of applicability
Min Temperature: 172.52 (K)
Max Temperature: 500 (K)
Max Density : 1691.1 (kg/m^3)
Max Pressure  : 6e+07 (Pa)

Thermal conductivity Range of applicability
Min Temperature: 172.52 (K)
Max Temperature: 500 (K)
Max Density : 1691.1 (kg/m^3)
Max Pressure  : 6e+07 (Pa)

Viscosity Range of applicability
Min Temperature: 172.52 (K) Max Temperature: 500 (K)
Max Density : 1692.3 (kg/m^3)
Max Pressure  : 6e+07 (Pa)
```

3. If you would like to model flow in the liquid phase, then this needs to be specified in the `set-phase` command. Note that the default phase is vapor, so if you do not go through this step, vapor is assumed. In addition, if the flow conditions do not permit liquid to exist, a vapor calculation will be performed instead.

```
  > define/user-defined/real-gas-models/set-phase
Select vapor phase (else liquid)?[yes]
```

Important

For mixture flows, not all combinations of species mixtures are allowed. This could be due to lack of data for one or more binary pairs. In such situations an error message generated by NIST will be returned and displayed on the ANSYS FLUENT console, and no real gas material is allowed to be created. In some combinations the mixing data will be estimated, a warning message will be displayed on the ANSYS FLUENT console and the mixture material allowed to be created.

8.16.4.4. Solution Strategies and Considerations for NIST Real Gas Model Simulation

The flow modeling of NIST real-gas flow is much more complex and challenging than simple ideal-gas flow. Therefore, you should expect the solution to converge at much slower rate with real-gas flow than when running ideal-gas flow. Also due to the complexity of the equations used in property evaluations, converging

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a solution with the real-gas model is in general done at much lower Courant values when you are using the density-based solver, or at much lower under-relaxation values if you are using the pressure-based solver. It is recommended that you first attempt to converge your solution using first-order discretization, then switch to second-order discretizations and re-iterate to convergence.

The real-gas properties in NIST are defined within a limited/bounded range. It is important that the flow conditions you are prescribing fall within the range of the database. It is possible that you specify flow at a state that is physically valid but otherwise not defined in the database. In this situation the solution will diverge or immediately generate an error message on the ANSYS FLUENT console as soon as the state crosses the limit of the database. In some instances, the actual converged state is just within the bounded defined database but only transitory outside the range. In this situation the divergence can be avoided by lowering the Courant value or under-relaxation factors so a less aggressive convergence rate is adapted.

Finally, if you attempt to initialize the flow from an inlet flow conditions and an error message is generated from one of the property routines, then this is an indicator that the flow conditions you have specified is not defined within the range of the database.

8.16.4.4.1. Writing Your Case File

When you save your completed real gas model to a case file, the linkage to the shared library containing real gas properties will be saved to the case file (along with property data for the material you selected in the NIST real gas model). Consequently, whenever you read your case file in a later session, ANSYS FLUENT will load and report this information to the console during the read process.

8.16.4.4.2. Postprocessing

All postprocessing functions properly report and display the current thermodynamic and transport properties of the selected real gas model. The thermodynamic and transport properties controlled by the NIST real gas model include the following:

- density
- enthalpy
- entropy
- molecular weight
- molecular viscosity
- sound speed
- specific heat
- thermal conductivity In addition to the properties listed above, you can also report
- compressibility factor
- any quantities that are derived from the properties listed above (e.g., total quantities, ratio of specific heats)

8.16.5. The User-Defined Real Gas Model

The user-defined real gas model (UDRGM) has been developed to allow you to write your own custom real gas model to fit your particular modeling needs. It also allows you to simulate a single-species flow, multiple-species mixture flow, multiphase flow, or volumetric reactions.

The following limitations exist for the User-Defined real gas model:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.5.1. Limitations of the User-Defined Real Gas Model

- You cannot include more that one User Defined Real Gas material (fluid or mixture) in the same problem. However, you can use other materials together with real gas in your simulation.
- When you are using the User Defined Real Gas model, the materials defined in your real gas UDF will appear in the *Create/Edit Materials Dialog Box* (p. 1736) with the name **real-gas-fluid** or **real-gas-mixture** and all physical and thermodynamic property inputs disabled. Use the *Create/Edit Materials Dialog Box* (p. 1736) to define or modify:
  - Mass diffusivity property in the real-gas-mixture material if you are modeling multi-component flow.
  - Chemical reactions in the real-gas-mixture if you are modeling reacting flow.
  - Radiation properties for the real-gas-mixture or the real-gas-fluid materials if you are modeling radiation.
  - Properties of materials other than the User Defined **real-gas-mixture** or **real-gas-fluid** materials.

  Note

  If you are using the User-Defined Real Gas model together with other materials from ANSYS FLUENT's property database in dispersed phase or multiphase calculations, take care to use the same reference temperature as in ANSYS FLUENT in your real-gas UDF. The reference temperature in ANSYS FLUENT is 298.15 K.

- Pressure-inlets, mass flow-inlets, and pressure-outlets are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions should not be used with the real gas models.
- The User Defined Real Gas model can be used with the Eulerian multiphase models.

  Note

  Note the following restrictions:
  - Only one phase can be a real gas.
  - Only **constant-rate** and **user-defined Mass Transfer Mechanisms** are available.

- The User Defined Real Gas model is compatible with the Lagrangian Dispersed Phase Models. Please refer to *Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models* (p. 465) for guidelines and restrictions of this approach.
- The real gas models cannot be used with the non-premixed, partially premixed, and composition PDF transport combustion models. Chemical reactions can however be modeled with the finite rate and eddy dissipation models. Please note that the **Dimension Reduction** model is not available with the real gas models.

The UDRGM requires a library of functions written in the C programming language. Moreover, there are certain coding requirements that need to be followed when writing these functions. Sample real gas function libraries are provided to assist you in writing your own UDRGM. When UDRGM functions are compiled, they will be grouped in a shared library which later will be loaded and linked with the ANSYS FLUENT executable. The procedure for using the UDRGM is defined as follows:

1. Define the real gas equation of state and all related thermodynamic and transport property equations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Create a C source code file that conforms to the format defined in this section.
3. Start ANSYS FLUENT and set up your case file in the usual way.
4. Compile your UDRGM C library and build a shared library file (you can use the available compiled UDF utilities in either the graphical user interface or the text command interface).
5. Load your newly created UDRGM library via the text command menu:

   If a single-species UDRGM to be used, then the text command menu is:

   ```
   > define/user-defined/real-gas-models/user-defined-real-gas-model

     use user defined real gas? [no] yes
   ```

   On the other hand, if you are simulating multiple-species UDRGM flow, then the text command menu to use is:

   ```
   > define/user-defined/real-gas-models/user-defined-multispecies-real-gas-model

     use user multispecies defined real gas? [no]yes
   ```

   Upon activating the UDRGM, the function library will now supply the fluid material properties for your case.
6. You can simulate volumetric reactions with your real gas model using the *Species Model Dialog Box* (p. 1681), or the text interface (`define/models/species/volumetric-reactions`?).

   You can access the **Species Model** by selecting **Models** from the navigation pane and double-clicking **Species** in the task page.

   In the *Species Model Dialog Box* (p. 1681)

   - Enable **Species Transport** under **Model**.
   - Enable **Volumetric** under **Reactions**.
   - Select the appropriate **Turbulence-Chemistry Interaction** option.
   - Set up the reaction by clicking the **Edit...** button for the **real-gas-mixture Mixture Material**.

   Important

   Note that the fluid materials and their properties, appearing in the *Create/Edit Materials Dialog Box* (p. 1736), are the ones defined in your real gas UDF. You cannot modify the materials via this dialog box, however, you can set up the volumetric reaction. If you would like to modify the mixture materials and their properties, this should be done in the real gas UDF. The volumetric reactions for your real gas mixture are defined in the same way as for any ANSYS FLUENT mixture. For details, refer to *Defining Reactions* (p. 809).

   Alternatively, the chemical reactions can be set up using the `define/models/species` and `define/materials` text command.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the chemical reactions should be activated after your real gas UDF has been built and loaded. It is also recommended to test and validate your real gas UDF, running the cold flow calculation prior to attempting to solve the reacting flow. Also, make sure that the applicability range of the real gas functions in your UDF fully covers the temperature and pressure range of the reacting flow calculation.

7. Run your calculation.

When using the UDRGM the robustness of the solver and the speed of flow convergence will largely depend on the complexity of the material properties you have defined in your UDF. It is important to understand the operational range of the property functions you are coding so you can simulate the flow within that range.

8.16.5.2. Writing the UDRGM C Function Library

Creating a UDRGM C function library is reasonably straightforward; however, your code must make use of specific function names and macros, which will be described in detail below. The basic library requirements are as follows:

- The code must contain the `udf.h` file inclusion directive at the beginning of the source code. This allows the definitions for `DEFINE` macros and other ANSYS FLUENT functions to be accessible during the compilation process.
- The code must include at least one in the UDF's `DEFINE` functions (i.e. `DEFINE_ON_DEMAND`) to be able to use the compiled UDFs utility (see the sample UDRGM codes provided below).
- Any values that are passed to the solver by the UDRGM or returned by the solver to the UDRGM are assumed to be in SI units.
- You must use the principal set of functions listed below in your UDRGM library. These functions are the mechanism by which your thermodynamic property data are transferred to the ANSYS FLUENT solver. Note that `ANYNAME` can be any string of alphanumeric characters, and allows you to provide unique names to your library functions.

Function inputs from the ANSYS FLUENT solver consist of one or more of the following variables:

T = Temperature, K

p = Pressure, Pa

ρ = Density, kg/m^3

$Y_i[]$ = Species mass fraction

Important

$Y_i[]$: ANSYS FLUENT solver returns a value of `1.0` for $Y_i[]$ in single-species flows. For multiple-species flows, $Y_i[]$ is a vector array containing species mass fraction in an order defined by the user setup function.

The UDRGM function names and argument lists, followed by a short description of the function are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`void ANYNAME_error(int err, char *f, char *msg)`
prints error messages.

`void ANYNAME_Setup(Domain *domain, cxboolean vapor_phase, char *filename,int (*messagefunc)(char *format,...), void (*errorfunc)(char *format, ...))`
performs model setup and initialization. Can be used to read data and parameters related to your UDRGM. When writing UDFs for multiple-species, use this function to specify the number of species and the name of the species as shown in the multiple-species example. The boolean variable, `vapor_phase`, passes to your UDF the setting of the text-interface command \ `define/user-defined/real-gas-models/set-phase`.

`double ANYNAME_density(double T, double P, double yi[])`
returns the value of density as a function of temperature, pressure and species mass-fraction if applicable. This is your equation of state.

Important

Since this function is called numerous times during each solver iteration, it is important to make this function as numerically efficient as possible.

`double ANYNAME_specific_heat(double T, double Rho, double P, double yi[])`
returns the real gas specific heat at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_enthalpy(double T, double Rho, double P, double yi[])`
returns the enthalpy as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_entropy(double T, double Rho, double P, double yi[])`
returns the entropy as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_mw(double yi[])`
returns the fluid molecular weight.

`double ANYNAME_speed_of_sound(double T, double Rho, double P, double yi[])`
returns the value of speed of sound as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_viscosity(double T, double Rho, double P, double yi[])`
returns the value of dynamic viscosity as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_thermal_conductivity(double T, double Rho, double P, double yi[])`
returns the value of thermal conductivity as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_rho_t(double T, double Rho, double P, double yi[])`
returns the value of $\frac{d\rho}{dT}$ at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_rho_p(double T, double Rho, double P, double yi[])`
returns the value of $\frac{d\rho}{dp}$ at constant temperature as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

double ANYNAME_enthalpy_t(double T, double Rho, double P, double yi[])

returns the value of $\frac{dh}{dT}$ at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable. Note that by definition $dh/dt==c_p$, so this function should simply return the specific heat value.

double ANYNAME_enthalpy_p(double T, double Rho, double P, double yi[])

returns the value of $\frac{dh}{dp}$ at constant temperature as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_enthalpy_prime(double T,double Rho,double P,double yi[],double hi[])

returns the value of the mixture enthalpy as a function of temperature, density, absolute pressure, and species mass fraction. In addition, your UDF will need to set the elements of the double array `hi[]` to the enthalpy of each species, in the same order as they are referenced in the mass fraction array `yi[]`. Note that the enthalpy in the function `enthalpy_prime` is defined as the sum of sensible enthalpy plus species formation enthalpy, and you should make sure that its computation is consistent with the sensible enthalpy function ANYNAME_`enthalpy`. The function ANYNAME_`enthalpy_prime` is required for the calculation of the heat of reactions, if chemical reactions are being simulated. If you are not solving reacting flows, the function ANYNAME_`enthalpy_prime` can simply be omitted.

At the end of the code you must define a structure of type RGAS_`Function` whose members are pointers to the principal functions listed above. The structure is of type RGAS_`Function` and its name is `RealGas-FunctionList`.

Important

It is imperative that the sequence of function pointers shown below be followed. Otherwise, your real gas model will not load properly into the ANSYS FLUENT code.

```
UDF_EXPORT RGAS_Functions RealGasFunctionList =
{
 ANYNAME_Setup,                    /* Setup initialize */
 ANYNAME_density,                  /* density */
 ANYNAME_enthalpy,                 /* sensible enthalpy */
 ANYNAME_entropy,                  /* entropy */
 ANYNAME_specific_heat,            /* specific_heat */
 ANYNAME_mw,                       /* molecular_weight */
 ANYNAME_speed_of_sound,           /* speed_of_sound */
 ANYNAME_viscosity,                /* viscosity */
 ANYNAME_thermal_conductivity,     /* thermal_conductivity */
 ANYNAME_rho_t,                    /* drho/dT |const p */
 ANYNAME_rho_p,                    /* drho/dp |const T */
 ANYNAME_enthalpy_t,               /* dh/dT |const p */
 ANYNAME_enthalpy_p                /* dh/dp |const T */
 ANYNAME_enthalpy_prime            /* enthalpy */
};
```

If volumetric reactions are not being simulated, then the function ANYNAME_`enthalpy_prime` can be removed or ignored from the `RealGasFunctionList` structure described here.

The principal set of functions described are the only functions in the UDRGM that will be interacting directly with the ANSYS FLUENT code. In many cases, your model may require further functions that will be called from the principal function set. For example, when multiple-species real gas model UDFs are written, the principal functions will return the mixture thermodynamic properties based on some specified mixing-law. Therefore, you may want to add further functions that will return the thermodynamic properties for the in-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dividual species. These auxiliary functions will be called from the principal set of functions. See User-Defined Real Gas Models (UDRGM) in the UDF Manual for examples that clearly illustrate this strategy.

8.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File

This section presents the steps you will need to follow to compile your UDRGM C code and build a shared library file. This process requires the use of a C compiler. Most UNIX operating systems provide a C compiler as a standard feature. If you are using a PC, you will need to ensure that a C++ compiler is installed before you can proceed (e.g., Microsoft Visual C++, v6.0 or higher). To use the UDRGM you will need to first build the UDRGM library by compiling your UDRGM C code and then load the library into the ANSYS FLUENT code. The UDRGM shared library is built in the same way that the ANSYS FLUENT executable itself is built. Internally, a script called `Makefile` is used to invoke the system C compiler to build an object code library that contains the native machine language translation of your higher-level C source code. This shared library is then loaded into ANSYS FLUENT (either at runtime or automatically when a case file is read) by a process called *dynamic loading*. The object libraries are specific to the computer architecture being used, as well as to the particular version of the ANSYS FLUENT executable being run. The libraries must, therefore, be rebuilt any time ANSYS FLUENT is upgraded, when the computer's operating system level changes, or when the job is run on a different type of computer. The general procedure for compiling UDRGM C code is as follows:

- Place the UDRGM C code in the folder, i.e., where your case file resides.
- Launch ANSYS FLUENT.
- Read your case file into ANSYS FLUENT.
- You can now compile your UDRGM C code and build a shared library file using either the graphical interface or the text command interface.

Important

To build UDRGM library you will use the compiled UDF utilities. However, you will not use the UDF utilities to load the library. A separate loading area for the UDRGM library will be used.

8.16.5.3.1. Compiling the UDRGM Using the Graphical Interface

If the build is successful, then the compiled library will be placed in the appropriate architecture folder (e.g., `ntx86/2d`). By default the library name is `libudf.so` (`libudf.dll` on Windows).

More information on compiled UDFs and building libraries using the ANSYS FLUENT graphical user interface can be found in the UDF Manual.

8.16.5.3.2. Compiling the UDRGM Using the Text Interface

The UDRGM library can be compiled in the text command interface as follows:

- Select the menu item `define` → `user-defined` → `compiled-functions`.
- Select the `compile` option.
- Enter the compiled UDF library name.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The name given here is the name of the folder where the shared library (e.g., `libudf`) will reside. For example, if you press `<Enter>` then a folder should exist with the name `libudf`, and this folder will contain a library file called `libudf`. If, however, you type a new library name such as `myrealgas`, then a folder called `myrealgas` will be created and it will contain the library `libudf`.

- Continue on with the procedure when prompted.
- Enter the C source file names.

Important

Ideally you should place all of your functions into a single file. However, you can split them into separate files if desired.

- Enter the header file names, if applicable. If you do not have an extra header file then hit `<Enter>` when prompted.

ANSYS FLUENT will then start compiling the UDRGM C code and put it in the appropriate architecture folder.

Example:

```
> define/user-defined/compiled-functions
  load OR compile ? [load]  compile


  Compiled UDF library name: ["libudf"] my_lib


   Make sure that UDF source files are in the folder
   that contains your case and data files. If you have
   an existing libudf folder, please remove this
   folder to ensure that latest files are used.
  Continue?[yes] RETURN


  Give C-Source file names:
  First file name: [""] my_c_file.c RETURN


  Next file name: [""] RETURN


  Give header file names:
  First file name: [""] my_header_file.h RETURN
```

8.16.5.3.3. Loading the UDRGM Shared Library File

Load the UDRGM library:

- Go to the following menu item in the text command interface.

  `define` → `user-defined` → `real-gas-models`

- Select one of the following
  - `user-defined-real-gas-model` if you are modeling a single-species real gas fluid

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - `user-defined-multispecies-real-gas-model` if you are modeling a multiple-species fluid-mixture

- Turn on the real gas model.
  - For single-species:

```
use user defined real gas? [no]  yes
```

  - For multiple-species:

```
use multispecies user defined real gas? [no]  yes
```

ANSYS FLUENT will ask for the location of the user-defined real gas library. You can enter either the name of the folder where the UDRGM shared library is called or the entire path to the UDRGM shared library.

If the loading of the UDRGM library is successful you will see a message similar to the following:

```
Opening user-defined realgas library "RealgasLibraryname"...
Library "RealgasDirName/lnx86/2d/libudf.so" opened
Setting material "air" to a real-gas...
Loading Real-RealGasPrefexLable Library:
```

8.16.5.4. UDRGM Example: Ideal Gas Equation of State

This section describes an example of a user-defined real gas model. You can use this example as the basis for your own UDRGM code. In this simple example, the standard ideal gas equation of state is used in the UDRGM. See User-Defined Real Gas Models (UDRGM) in the UDF Manual for more examples of UDRGM functions, including multi-species real gas and reacting real-gas examples.

p = pressure

T = temperature

C_p = specific heat

H = enthalpy

S = entropy

ρ = density

c = speed of sound

R = universal gas constant/molecular weight

The ideal gas equation of state can be written in terms of pressure and temperature as

$$\rho = \frac{p}{RT} \tag{8–107}$$

The specific heat is defined to be constant C_p = 1006.42.

The enthalpy is, therefore, defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H = C_p T \tag{8–108}$$

and entropy is given by

$$S = C_p \log\left(\frac{T}{T_{ref}}\right) + R \log\left(\frac{p_{ref}}{p}\right) \tag{8–109}$$

where T_{ref} = 288.15 K and p_{ref} = 101325 Pa

The speed of sound is simply defined as

$$c = \sqrt{T C_p \frac{R}{\left(C_p - R\right)}} \tag{8–110}$$

The density derivatives are:

$$\left(\frac{d\rho}{dp}\right)_T = \frac{1}{RT} \tag{8–111}$$

$$\left(\frac{d\rho}{dT}\right)_p = -\frac{p}{RT^2} = -\frac{\rho}{T} \tag{8–112}$$

The enthalpy derivatives are:

$$\left(\frac{dH}{dT}\right)_p = C_p \tag{8–113}$$

$$\left(\frac{dH}{dp}\right)_T = \frac{C_p}{\rho R}\left[1 - \frac{p}{\rho}\frac{d\rho}{dp}\right] = 0 \tag{8–114}$$

When you activate the real gas model and load the library successfully into ANSYS FLUENT, you will be using the equation of state and other fluid properties from this library rather than the one built into the ANSYS FLUENT code.

8.16.5.4.1. Ideal Gas UDRGM Code Listing

```
/*********************************************************************/
/* User Defined Real Gas Model :                                     */
/* For Ideal Gas Equation of State                                   */
/*                                                                   */
/*********************************************************************/
#include "udf.h"
#include "stdio.h"
#include "ctype.h"
#include "stdarg.h"

#define MW 28.966   /* molec. wt. for single gas (Kg/Kmol) */
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
#define RGAS (UNIVERSAL_GAS_CONSTANT/MW)

static int (*usersMessage)(char *,...);
static void (*usersError)(char *,...);


DEFINE_ON_DEMAND(I_do_nothing)
{
   /* This is a dummy function to allow us to use */
   /* the Compiled UDFs utility      */
}


void IDEAL_error(int err, char *f, char *msg)
{
 if (err)
   usersError("IDEAL_error (%d) from function: %s\n%s\n",err,f,msg);
}

void IDEAL_Setup(Domain *domain, cxboolean vapor_phase, char *filename,
    int (*messagefunc)(char *format, ...),
    void (*errorfunc)(char *format, ...))
{  /* Use this function for any initialization or model setups*/
     usersMessage = messagefunc;
     usersError  = errorfunc;
     usersMessage("\nLoading Real-Ideal Library: %s\n", filename);
}

double IDEAL_density(double Temp, double press, double yi[])
{
     double r = press/(RGAS*Temp); /* Density at Temp & press */
     return r;      /* (Kg/m^3) */
}

double IDEAL_specific_heat(double Temp, double density, double P, double yi[])
{
     double cp=1006.43;
     return cp;     /* (J/Kg/K) */
}

double IDEAL_enthalpy(double Temp, double density, double P, double yi[])
{
     double h=Temp*IDEAL_specific_heat(Temp, density, P, yi);
     return h;      /* (J/Kg) */
}

#define TDatum 288.15
#define PDatum 1.01325e5

double IDEAL_entropy(double Temp, double density, double P, double yi[])
{
 double s=IDEAL_specific_heat(Temp,density,P,yi)*log(fabs(Temp/TDatum))+
          RGAS*log(fabs(PDatum/P));
    return s;      /* (J/Kg/K) */
}


double IDEAL_mw(double yi[])
{
     return MW;     /* (Kg/Kmol) */
}

double IDEAL_speed_of_sound(double Temp, double density, double P, double yi[])
{
     double cp=IDEAL_specific_heat(Temp,density,P,yi);
     return sqrt(Temp*cp*RGAS/(cp-RGAS)); /* m/s */
}

double IDEAL_viscosity(double Temp, double density, double P, double yi[])
{
     double mu=1.7894e-05;
     return mu;     /* (Kg/m/s) */
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
}

double IDEAL_thermal_conductivity(double Temp, double density, double P,
              double yi[])
{
     double ktc=0.0242;
     return ktc;      /* W/m/K */
}
double IDEAL_rho_t(double Temp, double density, double P, double yi[])
{
     /* derivative of rho wrt. Temp at constant p */
     double rho_t=-density/Temp;
     return rho_t;     /* (Kg/m^3/K) */
}

double IDEAL_rho_p(double Temp, double density, double P, double yi[])
{
     /* derivative of rho wrt. pressure at constant T */
     double rho_p=1.0/(RGAS*Temp);
     return rho_p;    /* (Kg/m^3/Pa) */
}

double IDEAL_enthalpy_t(double Temp, double density, double P, double yi[])
{
     /* derivative of enthalpy wrt. Temp at constant p */
     return IDEAL_specific_heat(Temp, density, P, yi);
}

double IDEAL_enthalpy_p(double Temp, double density, double P, double yi[])
{
     /* derivative of enthalpy wrt. pressure at constant T  */
     /* general form dh/dp|T = (1/rho)*[ 1 + (T/rho)*drho/dT|p] */
     /* but for ideal gas dh/dp = 0        */
     return 0.0 ;
}

UDF_EXPORT RGAS_Functions RealGasFunctionList =
{
     IDEAL_Setup,                   /* initialize */
     IDEAL_density,                 /* density */
     IDEAL_enthalpy,                /* enthalpy */
     IDEAL_entropy,                 /* entropy */
     IDEAL_specific_heat,           /* specific_heat */
     IDEAL_mw,                      /* molecular_weight */
     IDEAL_speed_of_sound,          /* speed_of_sound */
     IDEAL_viscosity,               /* viscosity */
     IDEAL_thermal_conductivity,    /* thermal_conductivity */
     IDEAL_rho_t,                   /* drho/dT |const p */
     IDEAL_rho_p,                   /* drho/dp |const T */
     IDEAL_enthalpy_t,              /* dh/dT |const p  */
     IDEAL_enthalpy_p               /* dh/dp |const T  */
};
/*************************************************************/
```

8.16.5.5. Additional UDRGM Examples

You can find the following additional UDRGM examples in the UDF Manual:

- The Aungier Redlich Kwong equation of state for single component flow. See UDRGM Example: Redlich-Kwong Equation of State in the UDF Manual for details.
- A simple example of a multi-species real-gas model. See UDRGM Example: Multiple-Species Real Gas Model in the UDF Manual for details.
- A real gas model example with the Aungier Redlich Kwong equation of state, ideal gas mixing rules and volumetric reactions. See UDRGM Example: Real Gas Model with Volumetric Reactions in the UDF Manual for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: Modeling Basic Fluid Flow

This chapter describes the basic physical models that ANSYS FLUENT provides for fluid flow and the commands for defining and using them. Models for flows in moving zones (including sliding and dynamic meshes) are explained in *Modeling Flows with Moving Reference Frames* (p. 513), models for turbulence are described in *Modeling Turbulence* (p. 643), and models for heat transfer (including radiation) are presented in *Modeling Heat Transfer* (p. 691). An overview of modeling species transport and reacting flows is provided in *Modeling Species Transport and Finite-Rate Chemistry* (p. 799), details about models for species transport and reacting flows are described in *Modeling Species Transport and Finite-Rate Chemistry* (p. 799) – *Modeling a Composition PDF Transport Problem* (p. 911), and models for pollutant formation are presented in *Modeling Pollutant Formation* (p. 943). The discrete phase model is described in *Modeling Discrete Phase* (p. 1007), general multiphase models are described in *Modeling Multiphase Flows* (p. 1091), and the melting and solidification model is described in *Modeling Solidification and Melting* (p. 1201). For information on modeling porous media, porous jumps, and lumped parameter fans and radiators, see *Cell Zone and Boundary Conditions* (p. 205).

The information in this chapter is presented in the following sections:

9.1. User-Defined Scalar (UDS) Transport Equations
9.2. Periodic Flows
9.3. Swirling and Rotating Flows
9.4. Compressible Flows
9.5. Inviscid Flows

9.1. User-Defined Scalar (UDS) Transport Equations

For additional information, please see the following sections:

9.1.1. Introduction
9.1.2. UDS Theory
9.1.3. Setting Up UDS Equations in ANSYS FLUENT

9.1.1. Introduction

ANSYS FLUENT can solve the transport equation for an arbitrary, user-defined scalar (UDS) in the same way that it solves the transport equation for a scalar such as species mass fraction. Extra scalar transport equations may be needed in certain types of combustion applications or for example in plasma-enhanced surface reaction modeling. ANSYS FLUENT allows you to define additional scalar transport equations in your model in the *User-Defined Scalars Dialog Box* (p. 2113).

9.1.2. UDS Theory

9.1.2.1. Single Phase Flow

For an arbitrary scalar ϕ_k, ANSYS FLUENT solves the equation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial \rho \phi_k}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} k = 1, \ldots, N \tag{9–1}$$

where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term supplied by you for each of the N scalar equations. Note that Γ_k is defined as a tensor in the case of anisotropic diffusivity. The diffusion term is thus $\nabla \cdot \left(\Gamma_k \cdot \phi_k\right)$

For isotropic diffusivity, Γ_k could be written as $\Gamma_k I$ where I is the identity matrix.

For the steady-state case, ANSYS FLUENT will solve one of the three following equations, depending on the method used to compute the convective flux :

- If convective flux is *not* to be computed, ANSYS FLUENT will solve the equation

$$-\frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} k = 1, \ldots, N \tag{9–2}$$

  where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term supplied by you for each of the N scalar equations.

- If convective flux is to be computed with mass flow rate, ANSYS FLUENT will solve the equation

$$\frac{\partial}{\partial x_i}\left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} k = 1, \ldots, N \tag{9–3}$$

- It is also possible to specify a user-defined function to be used in the computation of convective flux. In this case, the user-defined mass flux is assumed to be of the form

$$F = \int_S \rho \vec{u} \cdot d\vec{S} \tag{9–4}$$

where $d\vec{S}$ is the face vector area.

Important

User-defined scalars in solid zones do not take into account the convective term with moving reference frames.

9.1.2.2. Multiphase Flow

For multiphase flows, ANSYS FLUENT solves transport equations for two types of scalars: *per phase* and *mixture*. For an arbitrary k scalar in *phase-1*, denoted by ϕ_l^k, ANSYS FLUENT solves the transport equation inside the volume occupied by *phase-l*

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial \alpha_l \rho_l \phi_l^k}{\partial t} + \nabla \cdot \left(\alpha_l \rho_l \vec{u}_l \phi_l^k - \alpha_l \Gamma_l^k \nabla \phi_l^k \right) = S_l^k k = 1, \ldots, N \quad \textbf{(9–5)}$$

where α_l, ρ_l, and $\vec{u}_l$ are the volume fraction, physical density, and velocity of *phase-l*, respectively. Γ_l^k and S_l^k are the diffusion coefficient and source term, respectively, which you will need to specify. In this case, scalar ϕ_l^k is associated only with one phase (*phase-l*) and is considered an individual field variable of *phase-l*.

The mass flux for *phase-l* is defined as

$$F_l = \int_S \alpha_l \rho_l \vec{u}_l \cdot d\vec{S} \quad \textbf{(9–6)}$$

If the transport variable described by scalar ϕ_l^k represents the physical field that is shared between phases, or is considered the same for each phase, then you should consider this scalar as being associated with a mixture of phases, ϕ^k. In this case, the generic transport equation for the scalar is

$$\frac{\partial \rho_m \phi^k}{\partial t} + \nabla \cdot \left(\rho_m \vec{u}_m \phi^k - \Gamma_m^k \nabla \phi^k \right) = S^{k}{}_m k = 1, \ldots, N \quad \textbf{(9–7)}$$

where mixture density ρ_m, mixture velocity $\vec{u}_m$, and mixture diffusivity for the scalar k Γ_m^k are calculated according to

$$\rho_m = \sum_l \alpha_l \rho_l \quad \textbf{(9–8)}$$

$$\rho_m \vec{u}_m = \sum_l \alpha_l \rho_l \vec{u}_l \quad \textbf{(9–9)}$$

$$F_m = \int_S rho_m \vec{u}_m \cdot d\vec{S} \quad \textbf{(9–10)}$$

$$\Gamma_m^k = \sum_l \alpha_l \Gamma_l^k \quad \textbf{(9–11)}$$

$$S_m^k = \sum_l S_l^k \quad \textbf{(9–12)}$$

To calculate mixture diffusivity, you will need to specify individual diffusivities for each material associated with individual phases.

Note that if the user-defined mass flux option is activated, then mass fluxes shown in *Equation 9–6* (p. 487) and *Equation 9–10* (p. 487) will need to be replaced in the corresponding scalar transport equations. For more information about the theoretical background of user-defined scalar transport equations, see User-Defined Scalar (UDS) Transport Equations in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.1.3. Setting Up UDS Equations in ANSYS FLUENT

ANSYS FLUENT allows you to define up to 50 user-defined scalar (UDS) transport equations in your model. The general scalar transport equation, Equation 1–8 in the Theory Guide, is shown below with the four terms (transient, flux, diffusivity, source) that you can customize. (*Figure 9.1* (p. 488)). You will define a UDS transport equation by setting the parameters for these four terms.

Figure 9.1 Generalized UDS Transport Equation

$$\underbrace{\frac{\partial \rho \phi_k}{\partial t}}_{\text{unsteady}} + \frac{\partial}{\partial x_i}\left(\underbrace{F_i \phi_k}_{\text{convection}} - \underbrace{\Gamma_k \frac{\partial \phi_k}{\partial x_i}}_{\text{diffusion}}\right) = \underbrace{S_{\phi_k}}_{\text{sources}} \qquad k = 1, \ldots, N_{scalars}$$

In addition, you can set boundary conditions for the variables within cells of a fluid or solid zone for a particular scalar equation. This is done by fixing the value of ϕ_k in *Figure 9.1* (p. 488). When ϕ_k is fixed in a given cell, the UDS scalar transport is not solved and the cell is not included when the residual sum is computed. Additionally, you can also specify custom boundary conditions in the mixture on all wall, inflow, and outflow boundaries on a per-scalar basis.

The procedures for setting up a user-defined scalar (UDS) equation for single-phase and multiphase flows are outlined below. Note that a significant difference between a UDS for a single-phase versus a multiphase application is that you will need to associate each UDS with its corresponding phase domain or mixture domain, depending on your application. If you supply UDFs for transient terms, convective fluxes, and sources, you will need to be aware that they are directly called from the phase or mixture domains, according to the scalar association settings.

See the UDF Manual for information on using UDFs to define scalar quantities.

9.1.3.1. Single Phase Flow

1. Specify the number of UDS equations you require in the *User-Defined Scalars Dialog Box* (p. 2113) (*Figure 9.2* (p. 489)).

   Define $\rightarrow$ **User-Defined** $\rightarrow$ **Scalars...**

   Important

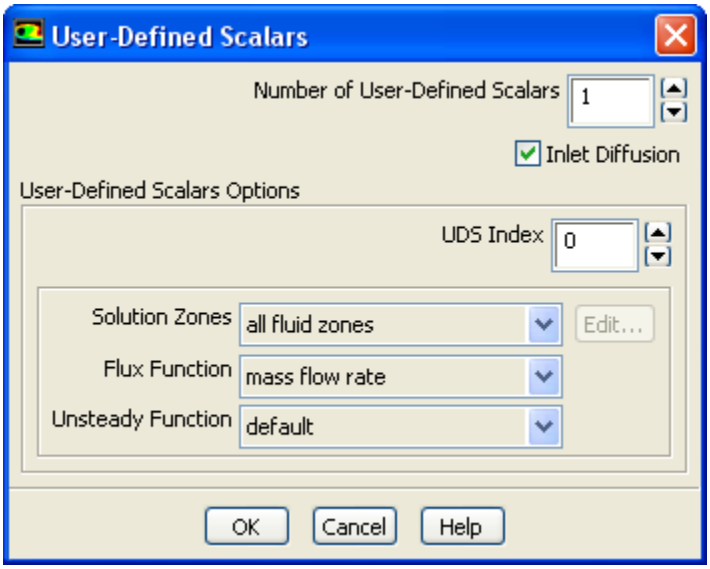
   The maximum number of user-defined scalar transport equations you can define is 50. ANSYS FLUENT assigns numbers to the equations starting with 0.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that ANSYS FLUENT assigns a default name for each scalar equation (`UserScalar0`, `User Scalar 1`, etc.). These labels will appear in graphics dialog boxes in ANSYS FLUENT. You can change them by means of a UDF. See the UDF Manual for details.

Figure 9.2 The User-Defined Scalars Dialog Box



2. Enable **Inlet Diffusion** if you want to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

3. Set the first user-defined scalar equation parameters by making sure that the **UDS Index** is set to 0.

   a. Specify the **Solution Zones** you want the scalar equation to be solved in as **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**. If you choose **selected zones**, click on the **Edit** button to view the list of zones you can select.

   b. Specify the **Flux Function** to be **none**, **mass flow rate**, or a user-defined function (UDF). The **Flux Function** determines how the convective flux is computed, which determines the equation that ANSYS FLUENT solves for the user-defined scalar. Selecting **none**, **mass flow rate**, or a user-defined function results in ANSYS FLUENT solving Equation 1–9, Equation 1–10, or Equation 1–11, respectively (in the Theory Guide). See the UDF Manual for details on flux UDFs.

   c. Specify the **Unsteady Function** to be **none**, **default**, or a user-defined function (UDF). Select **none** for a steady state solution and **default** if you want the transient term in Equation 1–8 in the Theory Guide). See the separate UDF Manual for details on unsteady UDFs.

   d. Repeat this process for each scalar equation by incrementing the **UDS Index**.

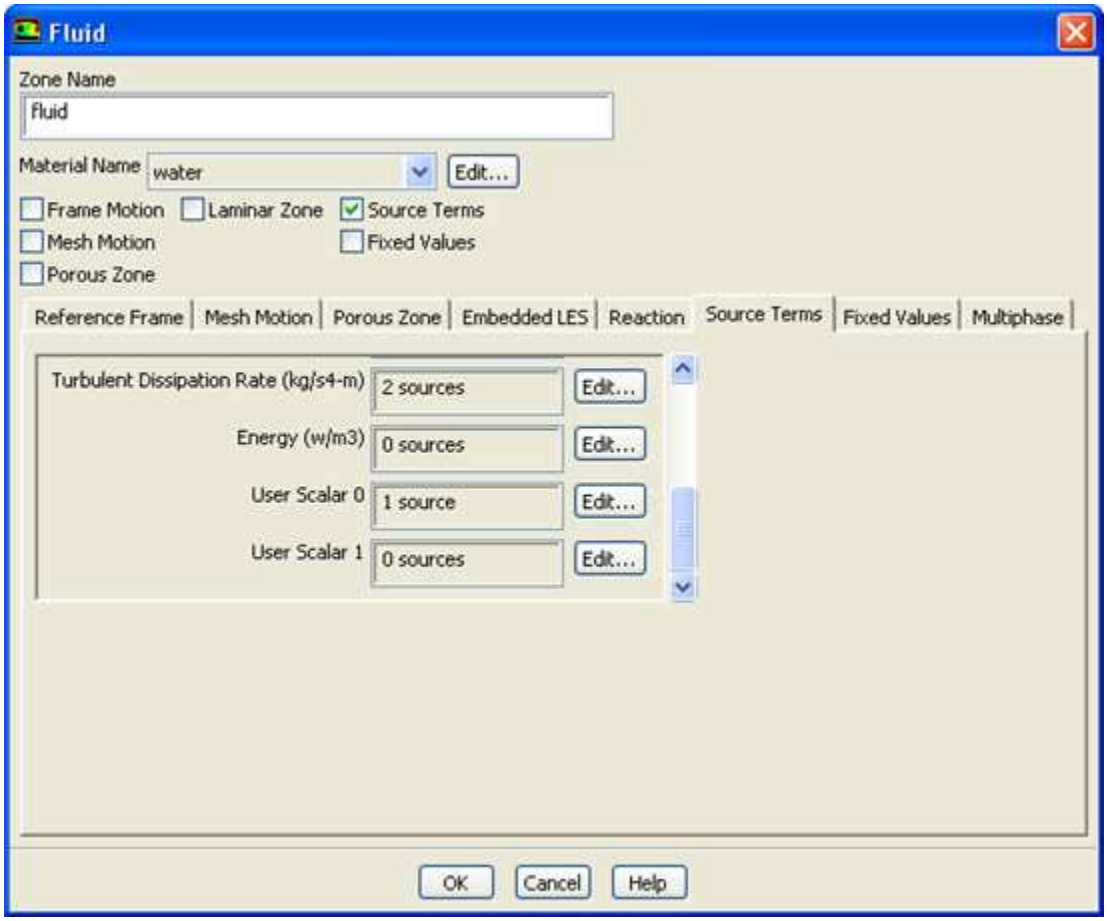
   e. Click **OK** when all user scalar equations have been defined.

4. To specify source term(s) for each of the N UDS equations, enable the **Source Terms** option in the **Fluid** or **Solid** dialog box (*Figure 9.3* (p. 490)) and click on the **Source Terms** tab. The source parameters will be displayed.

Cell Zone Conditions

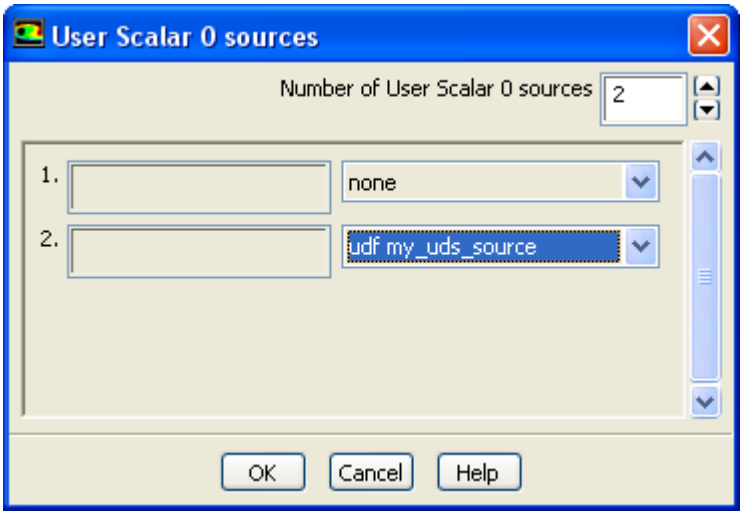
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.3 The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar



a. Specify the number of sources you require for each scalar equation by clicking on the **Edit...** button next to the scalar name (e.g., **User Scalar 0**). This will open the **User Scalar 0 sources** dialog box (*Figure 9.4* (p. 490)).

Figure 9.4 The User Scalar Sources Dialog Box



b. Specify the **Number of User Scalar Sources** for the scalar equation by incrementing the counter. Based on the value you have chosen, the sources will be added to the list in the dialog box.

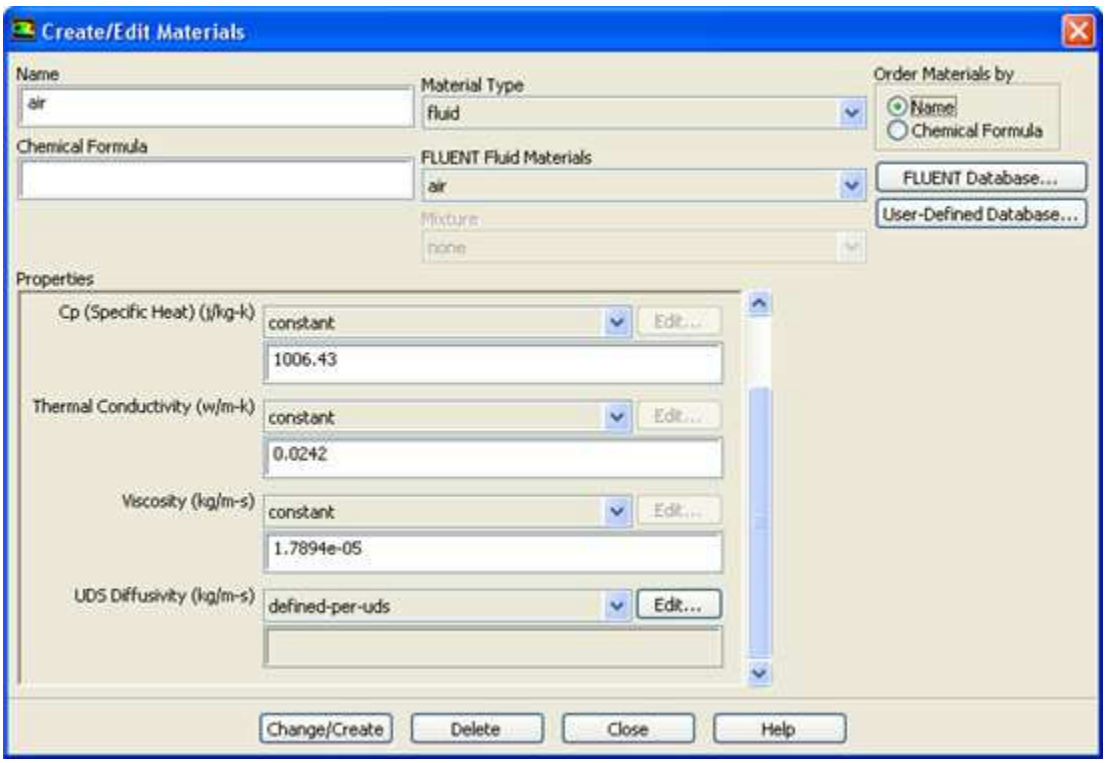
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify each source to be **none**, **constant**, or a user-defined function (UDF). For details on defining a UDF scalar source, see the UDF Manual. Click **OK** when you have specified all scalar sources.

5. To specify diffusivity for each of the N UDS equations, display the *Materials Task Page* (p. 1735) (*Figure 9.5* (p. 491)) and select either **defined-per-uds** (the default) or **user-defined** in the drop-down list for **UDS Diffusivity**.

Materials → **Create/Edit**

Figure 9.5 The Materials Dialog Box with Input for Diffusivity for UDS Equations



See *User-Defined Scalar (UDS) Diffusivity* (p. 427) for details on the different options available to you for defining diffusion coefficients.

6. To specify boundary conditions for the user-defined scalars on wall, inflow, and outflow boundaries, you can define a specific value or a specific flux for each scalar. A coupled boundary condition can be specified on two-sided walls for scalars that are to be solved in regions on both sides of the wall (i.e., scalars solved in both **fluid and solid** zones).

Boundary Conditions

a. In the **UDS** tab under **User Defined Scalar Boundary Condition**, select either **Specified Flux** or **Specified Value** in the drop-down list next to each scalar (e.g., **User Scalar 0**) for a boundary wall. For interior walls, select **Coupled Boundary** if the scalars are to be solved on both sides of a two-sided wall. Note that the **Coupled Boundary** option will only show up in the drop-down list if the scalar is defined in the **fluid and solid** zones in the *User-Defined Scalars Dialog Box* (p. 2113).

b. Under **User Defined Scalar Boundary Value**, enter a constant value or select a user-defined function from the drop-down list for each scalar. If you select **Specified Flux**, your input will be the value of the flux at the boundary (i.e., the negative of the term in parenthesis on the left hand side of Equation 1–9 in the Theory Guide) dot [as in the dot product of] n [as in the vector, n],

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where n is the normal into the domain). If you select **Specified Value**, your input will be the value of the scalar itself at the boundary. See the UDF Manual for information on using UDFs for UDS boundary conditions.

7. Set the solution parameters in the **Solution Controls** task page, specify an initial value for each UDS (as you do for all other scalar transport equations), and calculate a solution.

8. Examine the results using the usual postprocessing tools. In each postprocessing dialog box, the list of field variables will include the **User Defined Scalars...** category, which contains the value of each UDS and its diffusion coefficient (Γ_k in Equation 1–8, Equation 1–9, Equation 1–10, or Equation 1–11 (in the Theory Guide):

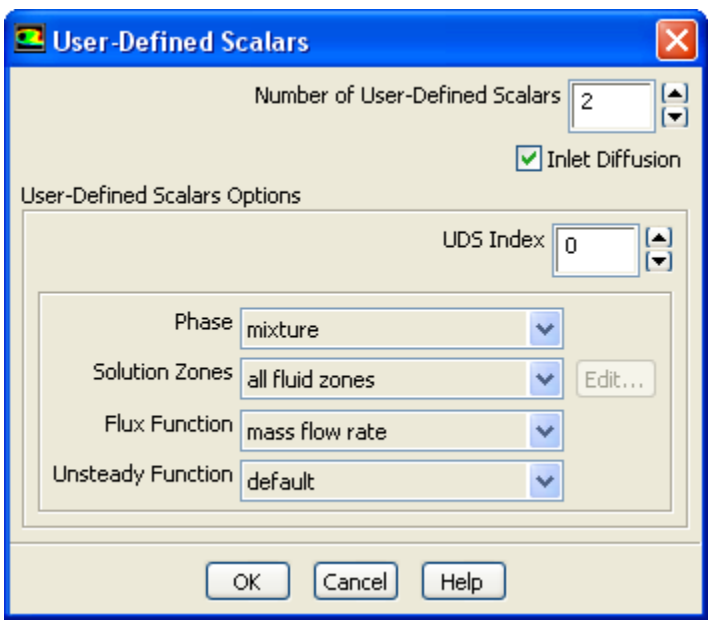
   - **User Scalar-n**
   - **Diffusion Coef. of Scalar-n**

9.1.3.2. Multiphase Flow

1. Specify the number of scalars in the*User-Defined Scalars Dialog Box* (p. 2113) (*Figure 9.6* (p. 492)).

   Define → **User-Defined** → **Scalars...**

   Figure 9.6 The User-Defined Scalars Dialog Box for a Multiphase Flow



Important

The maximum number of user-defined scalar transport equations you can define is 50. ANSYS FLUENT assigns numbers to the equations starting with 0. The default association type is set to `mixture` for all scalars.

Important

Note that ANSYS FLUENT assigns a default name for each scalar equation (`User Scalar 0`, `User Scalar 1`, etc.). These labels will appear in graphics dialog boxes in ANSYS FLUENT. You can change them by means of a UDF. See the UDF Manual for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Keep the default **Inlet Diffusion** enabled if you want to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.
3. Set the first user-defined scalar equation parameters by making sure that the **UDS Index** is set to 0.
   a. Select the **Phase** you want the scalar equation solved in as a primary phase, secondary phase, or the mixture.
   b. Specify the **Solution Zones** you want the scalar equation to be solved in as **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**. If you choose **selected zones**, click on the **Edit** button to view the list of zones you can select.
   c. Specify the **Flux Function** to **Unsteady Function** the same way as you would for a single phase flow (see above).
   d. Repeat this process for each scalar equation by incrementing the **UDS Index**.
   e. Click **OK** when all user scalar equations have been defined.
4. Specify source term(s) for each of the N UDS equations in the **Fluid** or **Solid** dialog box as described for a single phase flow (see above).
5. Specify boundary conditions for the user-defined scalars in the mixture on all wall, inflow, and outflow boundary as described for a single phase flow (see above).
6. Set the solution parameters, specify an initial value for each UDS (as you do for all other scalar transport equations), and calculate a solution.

9.2. Periodic Flows

Periodic flow occurs when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic flow can be modeled in ANSYS FLUENT. In the first type, no pressure drop occurs across the periodic planes. In the second type, a pressure drop occurs across translationally periodic boundaries, resulting in “fully-developed” or “streamwise-periodic” flow.

This section discusses streamwise-periodic flow. A description of no-pressure-drop periodic flow is provided in *Periodic Boundary Conditions* (p. 321), and a description of streamwise-periodic heat transfer is provided in *Modeling Periodic Heat Transfer* (p. 763).

Information about streamwise-periodic flow is presented in the following sections:

9.2.1. Overview and Limitations
9.2.2. User Inputs for the Pressure-Based Solver
9.2.3. User Inputs for the Density-Based Solvers
9.2.4. Monitoring the Value of the Pressure Gradient
9.2.5. Postprocessing for Streamwise-Periodic Flows

For more information about the theoretical background of periodic flows, see Periodic Flows in the Theory Guide.

9.2.1. Overview and Limitations

9.2.1.1. Overview

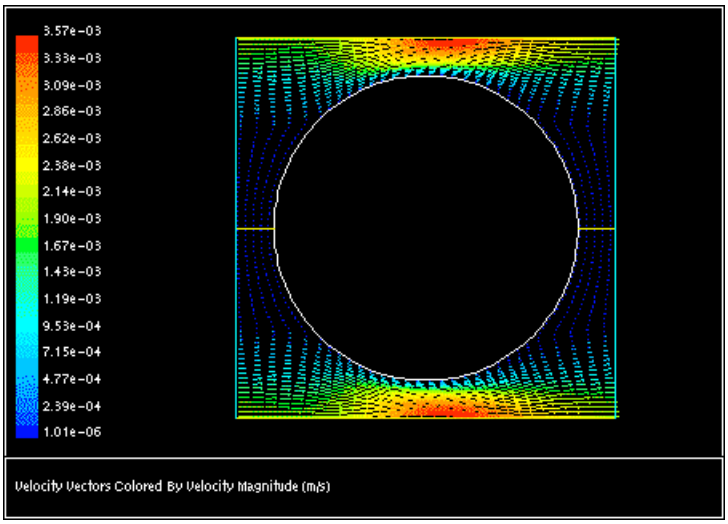
ANSYS FLUENT provides the ability to calculate streamwise-periodic—or “fully-developed”—fluid flow. These flows are encountered in a variety of applications, including flows in compact heat exchanger channels and flows across tube banks. In such flow configurations, the geometry varies in a repeating manner along the direction of the flow, leading to a periodic fully-developed flow regime in which the flow pattern repeats

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in successive cycles. Other examples of streamwise-periodic flows include fully-developed flow in pipes and ducts. These periodic conditions are achieved after a sufficient entrance length, which depends on the flow Reynolds number and geometric configuration.

Streamwise-periodic flow conditions exist when the flow pattern repeats over some length L, with a constant pressure drop across each repeating module along the streamwise direction. *Figure 9.7* (p. 494) depicts one example of a periodically repeating flow of this type which has been modeled by including a single representative module.

Figure 9.7 Example of Periodic Flow in a 2D Heat Exchanger Geometry



9.2.1.2. Limitations for Modeling Streamwise-Periodic Flow

The following limitations apply to modeling streamwise-periodic flow:

- The flow must be incompressible.
- When performing unsteady-state simulations with translational periodic boundary conditions, the specified pressure gradient is recommended.
- If one of the density-based solvers is used, you can specify only the pressure jump; for the pressure-based solver, you can specify either the pressure jump or the mass flow rate.
- No net mass addition through inlets/exits or extra source terms is allowed.
- Species can be modeled only if inlets/exits (without net mass addition) are included in the problem. Reacting flows are not permitted.
- Steady particle tracks can be modeled only if the particles have a possibility to leave the domain without generating incomplete trajectories.
- While Eulerian multiphase can be modeled with translational periodic boundary conditions, you cannot use the mass flow rate specification method. However, you can specify a constant pressure gradient.

9.2.2. User Inputs for the Pressure-Based Solver

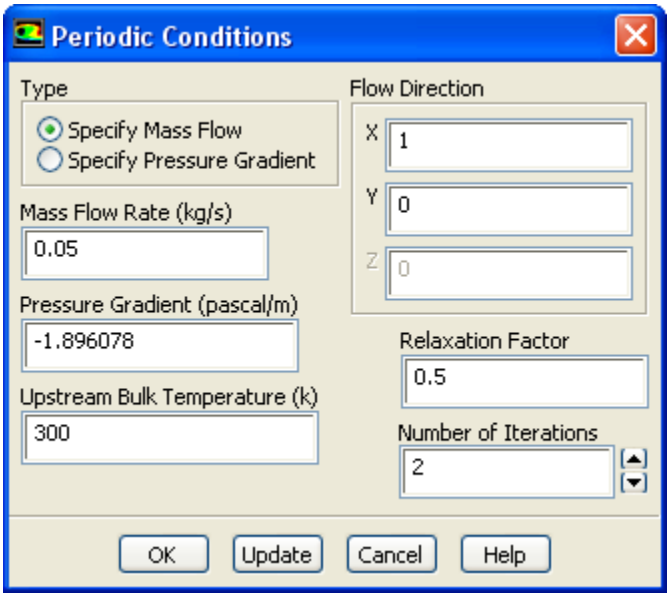
If you are using the pressure-based solver, in order to calculate a spatially periodic flow field with a specified mass flow rate or pressure derivative, you must first create a mesh with translationally periodic boundaries that are parallel to each other and equal in size. You can specify translational periodicity in the *Periodic Conditions Dialog Box* (p. 1874), as described in *Periodic Boundary Conditions* (p. 321). (If you need to create periodic boundaries, see *Creating Conformal Periodic Zones* (p. 191).)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the *Periodic Conditions Dialog Box* (p. 1874) which is opened from the *Boundary Conditions Task Page* (p. 1812), you will complete the following inputs after the mesh has been read into ANSYS FLUENT (*Figure 9.8* (p. 495)):

Boundary Conditions → **Periodic Conditions...**

Figure 9.8 The Periodic Conditions Dialog Box



1. Select either the specified mass flow rate (**Specify Mass Flow**) option or the specified pressure gradient (**Specify Pressure Gradient**) option. For most problems, the mass flow rate across the periodic boundary will be a known quantity; for others, the mass flow rate will be unknown, but the pressure gradient (β in Equation 1–22, in the Theory Guide) will be a known quantity.

2. Specify the mass flow rate and/or the pressure gradient (β in Equation 1–22, in the Theory Guide):

   - If you selected the **Specify Mass Flow** option, enter the desired value for the **Mass Flow Rate**. You can also specify an initial guess for the **Pressure Gradient**, but this is not required.

     Important

     For axisymmetric problems, the mass flow rate is per 2π radians.

   - If you selected the **Specify Pressure Gradient** option, enter the desired value for **Pressure Gradient**.

3. Define the flow direction by setting the **X,Y,Z** (or **X,Y** in 2D) point under **Flow Direction**. The flow will move in the direction of the vector pointing from the origin to the specified point. The direction vector must be parallel to the periodic translation direction or its opposite.

4. If you chose in step 1 to specify the mass flow rate, set the parameters used for the calculation of β. These parameters are described in detail below.

After completing these inputs, you can solve the periodic velocity field to convergence.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.2.2.1. Setting Parameters for the Calculation of β

If you choose to specify the mass flow rate, ANSYS FLUENT will need to calculate the appropriate value of the pressure gradient β. You can control this calculation by specifying the **Relaxation Factor** and the **Number of Iterations**, and by supplying an initial guess for β. All of these inputs are entered in the *Periodic Conditions Dialog Box* (p. 1874).

The **Number of Iterations** sets the number of sub-iterations performed on the correction of β in the pressure correction equation. Because the value of β is not known a priori, it must be iterated on until the **Mass Flow Rate** that you have defined is achieved in the computational model. This correction of β occurs in the pressure correction step of the SIMPLE, SIMPLEC, or PISO algorithm. A correction to the current value of β is calculated based on the difference between the desired mass flow rate and the actual one. The sub-iterations referred to here are performed within the pressure correction step to improve the correction for β before the pressure correction equation is solved for the resulting pressure (and velocity) correction values. The default value of 2 sub-iterations should suffice in most problems, but can be increased to help speed convergence. The **Relaxation Factor** is an under-relaxation factor that controls convergence of this iteration process.

You can also speed up convergence of the periodic calculation by supplying an initial guess for β in the **Pressure Gradient** field. Note that the current value of β will be displayed in this field if you have performed any calculations. To update the **Pressure Gradient** field with the current value at any time, click on the **Update** button.

9.2.3. User Inputs for the Density-Based Solvers

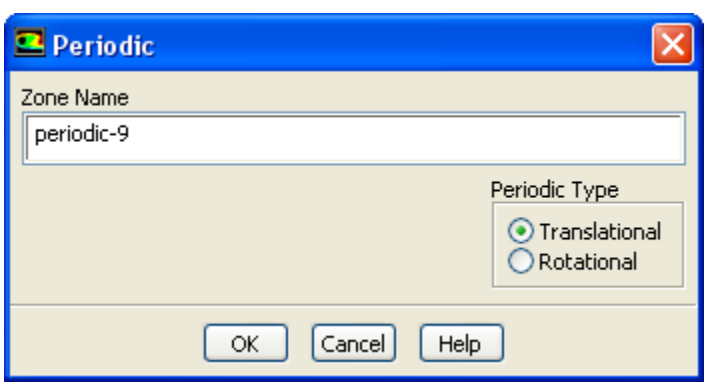
If you are using one of the density-based solvers, in order to calculate a spatially periodic flow field with a specified pressure jump, you must first create a mesh with translationally periodic boundaries that are parallel to each other and equal in size. (If you need to create periodic boundaries, see *Creating Conformal Periodic Zones* (p. 191).)

Then, follow the steps below:

1. In the *Periodic Dialog Box* (p. 1842) (*Figure 9.9* (p. 496)), which is opened from the **Boundary Conditions** task page, indicate that the periodicity is **Translational** (the default).

   Boundary Conditions → **Periodic** → **Edit...**

   Figure 9.9 The Periodic Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Also in the *Periodic Dialog Box* (p. 1842), set the **Periodic Pressure Jump** (Δp in Equation 1–21 in the Theory Guide).

After completing these inputs, you can solve the periodic velocity field to convergence.

9.2.4. Monitoring the Value of the Pressure Gradient

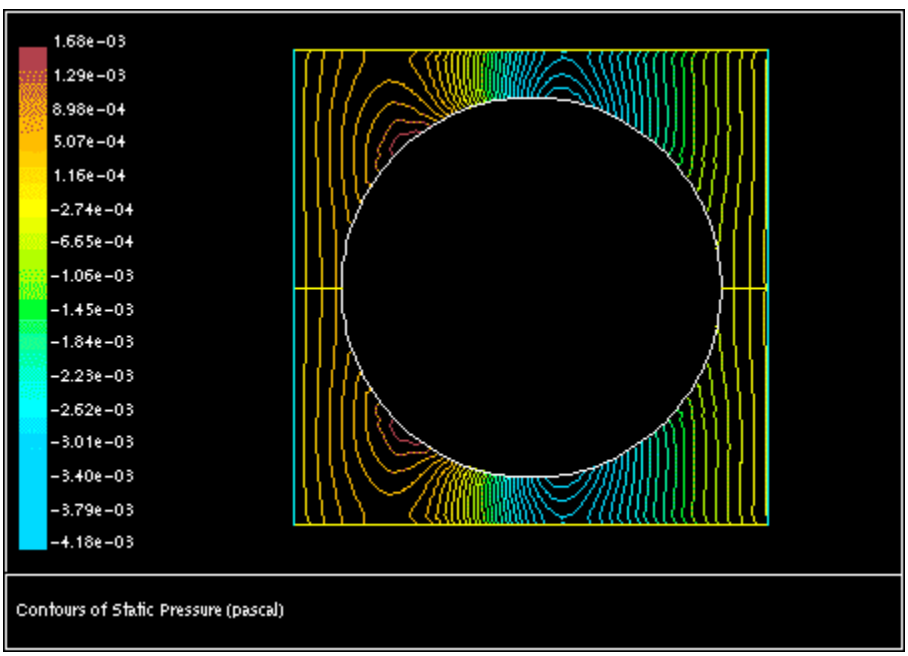
If you have specified the mass flow rate, you can monitor the value of the pressure gradient β during the calculation using the *Statistic Monitors Dialog Box* (p. 1917). Select **per/pr-grad** as the variable to be monitored. See *Monitoring Statistics* (p. 1277) for details about using this feature.

9.2.5. Postprocessing for Streamwise-Periodic Flows

For streamwise-periodic flows, the velocity field should be completely periodic. If a density-based solver is used to compute the periodic flow, the pressure field reported will be the actual pressure p (which is not periodic). If the pressure-based solver is used, the pressure field reported will be the periodic pressure field $\tilde{p}(\vec{r})$ of Equation 1–22, in the Theory Guide. *Figure 9.10* (p. 497) displays the periodic pressure field in the geometry of *Figure 9.7* (p. 494).

If you specified a mass flow rate and had ANSYS FLUENT calculate the pressure gradient, you can check the pressure gradient in the streamwise direction (β) by looking at the current value for **Pressure Gradient** in the *Periodic Conditions Dialog Box* (p. 1874).

Figure 9.10 Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry



9.3. Swirling and Rotating Flows

Many important engineering flows involve swirl or rotation and ANSYS FLUENT is well-equipped to model such flows. Swirling flows are common in combustion, with swirl introduced in burners and combustors in order to increase residence time and stabilize the flow pattern. Rotating flows are also encountered in turbomachinery, mixing tanks, and a variety of other applications.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Information about rotating and swirling flows is provided in the following subsections:

9.3.1. Overview of Swirling and Rotating Flows
9.3.2. Turbulence Modeling in Swirling Flows
9.3.3. Mesh Setup for Swirling and Rotating Flows
9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation

For more information about the theoretical background of swirling and rotating flows, see Swirling and Rotating Flows in the Theory Guide.

When you begin the analysis of a rotating or swirling flow, it is essential that you classify your problem into one of the following five categories of flow:

- axisymmetric flows with swirl or rotation
- fully three-dimensional swirling or rotating flows
- flows requiring a moving reference frame
- flows requiring multiple moving reference frames or mixing planes
- flows requiring sliding meshes

Modeling and solution procedures for the first two categories are presented in this section. The remaining three, which all involve "moving zones", are discussed in *Modeling Flows with Moving Reference Frames* (p. 513).

9.3.1. Overview of Swirling and Rotating Flows

9.3.1.1. Axisymmetric Flows with Swirl or Rotation

Your problem may be axisymmetric with respect to geometry and flow conditions but still include swirl or rotation. In this case, you can model the flow in 2D (i.e., solve the axisymmetric problem) and include the prediction of the circumferential (or swirl) velocity. It is important to note that while the assumption of axisymmetry implies that there are no circumferential gradients in the flow, there may still be non-zero swirl velocities.

9.3.1.1.1. Momentum Conservation Equation for Swirl Velocity

The tangential momentum equation for 2D swirling flows may be written as

$$\frac{\partial}{\partial t}(\rho w)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho u w)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v w)=\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial w}{\partial x}\right]+\frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{w}{r}\right)\right]-\rho\frac{vw}{r} \qquad \textbf{(9–13)}$$

where x is the axial coordinate, r is the radial coordinate, u is the axial velocity, v is the radial velocity, and w is the swirl velocity.

9.3.1.2. Three-Dimensional Swirling Flows

When there are geometric changes and/or flow gradients in the circumferential direction, your swirling flow prediction requires a three-dimensional model. If you are planning a 3D ANSYS FLUENT model that includes swirl or rotation, you should be aware of the setup constraints listed in *Coordinate System Restrictions* (p. 500). In addition, you may wish to consider simplifications to the problem which might reduce it to an equivalent axisymmetric problem, especially for your initial modeling effort. Because of the complexity of swirling flows, an initial 2D study, in which you can quickly determine the effects of various modeling and design choices, can be very beneficial.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

For 3D problems involving swirl or rotation, there are no special inputs required during the problem setup and no special solution procedures. Note, however, that you may want to use the cylindrical coordinate system for defining velocity-inlet boundary condition inputs, as described in *Defining the Velocity* (p. 266). Also, you may find the gradual increase of the rotational speed (set as a wall or inlet boundary condition) helpful during the solution process. This is described for axisymmetric swirling flows in *Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed* (p. 503).

9.3.1.3. Flows Requiring a Moving Reference Frame

If your flow involves a rotating boundary which moves through the fluid (e.g., an impeller blade or a grooved or notched surface), you will need to use a moving reference frame to model the problem. Such applications are described in detail in *Introduction* (p. 513). If you have more than one rotating boundary (e.g., several impellers in a row), you can use multiple reference frames (described in *The Multiple Reference Frame Model* (p. 522)) or mixing planes (described in *The Mixing Plane Model* (p. 523)).

9.3.2. Turbulence Modeling in Swirling Flows

If you are modeling turbulent flow with a significant amount of swirl (e.g., cyclone flows, swirling jets), you should consider using one of ANSYS FLUENT's advanced turbulence models: the RNG k- ε model, realizable k- ε model, or Reynolds stress model. The appropriate choice depends on the strength of the swirl, which can be gauged by the swirl number. The swirl number is defined as the ratio of the axial flux of angular momentum to the axial flux of axial momentum:

$$S = \frac{\int rw\vec{v} \cdot d\vec{A}}{\overline{R} \int u\vec{v} \cdot d\vec{A}} \qquad \textbf{(9–14)}$$

where $\overline{R}$ is the hydraulic radius.

For flows with weak to moderate swirl ($S < 0.5$), both the RNG k- ε model and the realizable k- ε model yield appreciable improvements over the standard k- ε model. See RNG k- ε Model and Realizable k- ε Model *Swirl Modification* (p. 674) for details about these models.

For highly swirling flows ($S > 0.5$), the Reynolds stress model (RSM) is strongly recommended. The effects of strong turbulence anisotropy can be modeled rigorously only by the second-moment closure adopted in the RSM. See Reynolds Stress Model (RSM) *Steps in Using a Turbulence Model* (p. 655) for details about this model.

For swirling flows encountered in devices such as cyclone separators and swirl combustors, near-wall turbulence modeling is quite often a secondary issue at most. The fidelity of the predictions in these cases is mainly determined by the accuracy of the turbulence model in the core region. However, in cases where walls actively participate in the generation of swirl (i.e., where the secondary flows and vortical flows are generated by pressure gradients), non-equilibrium wall functions can often improve the predictions since they use a law of the wall for mean velocity sensitized to pressure gradients. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for additional details about near-wall treatments for turbulence.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.3. Mesh Setup for Swirling and Rotating Flows

9.3.3.1. Coordinate System Restrictions

Recall that for an axisymmetric problem, the axis of rotation must be the x axis and the mesh must lie on or above the $y=0$ line.

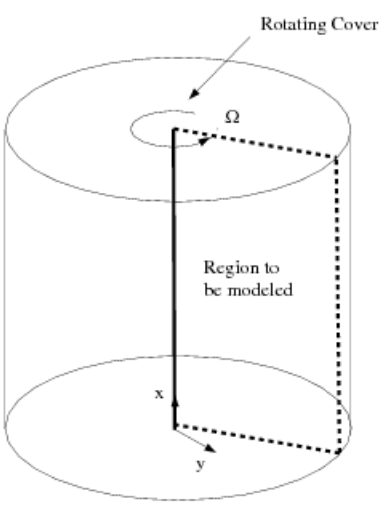
9.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows

In addition to the setup constraint described above, you should be aware of the need for sufficient resolution in your mesh when solving flows that include swirl or rotation. Typically, rotating boundary layers may be very thin, and your ANSYS FLUENT model will require a very fine mesh near a rotating wall. In addition, swirling flows will often involve steep gradients in the circumferential velocity (e.g., near the centerline of a free-vortex type flow), and thus require a fine mesh for accurate resolution.

9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation

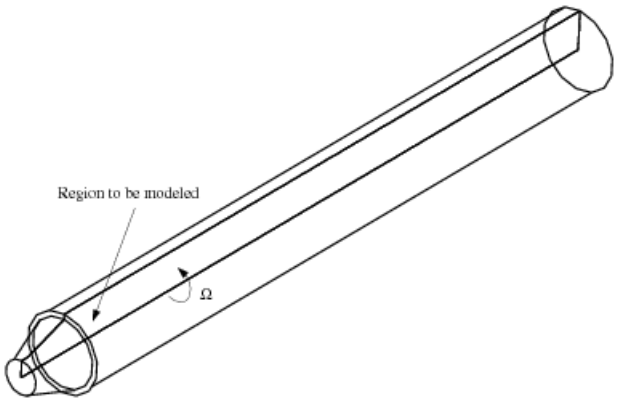
As discussed in *Overview of Swirling and Rotating Flows* (p. 498), you can solve a 2D axisymmetric problem that includes the prediction of the circumferential or swirl velocity. The assumption of axisymmetry implies that there are no circumferential gradients in the flow, but that there may be non-zero circumferential velocities. Examples of axisymmetric flows involving swirl or rotation are depicted in *Figure 9.11* (p. 500) and *Figure 9.12* (p. 501).

Figure 9.11 Rotating Flow in a Cavity



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.12 Swirling Flow in a Gas Burner



9.3.4.1. Problem Setup for Axisymmetric Swirling Flows

For axisymmetric problems, you will need to perform the following steps during the problem setup procedure. (Only those steps relevant specifically to the setup of axisymmetric swirl/rotation are listed here. You will need to set up the rest of the problem as usual.)

1. Activate solution of the momentum equation in the circumferential direction by turning on the **Axisymmetric Swirl** option for **Space** in the **General** task page.

   General → **Axisymmetric**

2. Define the rotational or swirling component of velocity, $r\Omega$, at inlets or walls.

   Boundary Conditions

   Important

   Remember to use the axis boundary type for the axis of rotation.

The procedures for input of rotational velocities at inlets and at walls are described in detail in *Defining the Velocity* (p. 266) and *Velocity Conditions for Moving Walls* (p. 302).

9.3.4.2. Solution Strategies for Axisymmetric Swirling Flows

The difficulties associated with solving swirling and rotating flows are a result of the high degree of coupling between the momentum equations, which is introduced when the influence of the rotational terms is large. A high level of rotation introduces a large radial pressure gradient which drives the flow in the axial and radial directions. This, in turn, determines the distribution of the swirl or rotation in the field. This coupling may lead to instabilities in the solution process, and you may require special solution techniques in order to obtain a converged solution. Solution techniques that may be beneficial in swirling or rotating flow calculations include the following:

- (Pressure-based segregated solver only) Use the PRESTO! scheme (enabled in the **Pressure** list for **Spatial Discretization** in the *Solution Methods Task Page* (p. 1900)), which is well-suited for the steep pressure gradients involved in swirling flows.
- Ensure that the mesh is sufficiently refined to resolve large gradients in pressure and swirl velocity.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- (Pressure-based solver only) Change the under-relaxation parameters on the velocities, perhaps to 0.3–0.5 for the radial and axial velocities and 0.8–1.0 for swirl.
- (Pressure-based solver only) Use a sequential or step-by-step solution procedure, in which some equations are temporarily left inactive (see below).
- If necessary, start the calculations using a low rotational speed or inlet swirl velocity, increasing the rotation or swirl gradually in order to reach the final desired operating condition (see below).

See *Using the Solver* (p. 1207) for details on the procedures used to make these changes to the solution parameters. More details on the step-by-step procedure and on the gradual increase of the rotational speed are provided below.

9.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows

Often, flows with a high degree of swirl or rotation will be easier to solve if you use the following step-by-step solution procedure, in which only selected equations are left active in each step. This approach allows you to establish the field of angular momentum, then leave it fixed while you update the velocity field, and then finally to couple the two fields by solving all equations simultaneously.

Important

Since the density-based solvers solve all the flow equations simultaneously, the following procedure applies only to the pressure-based solver.

In this procedure, you will use the **Equations...** button in the *Solution Controls Task Page* (p. 1902) to turn individual transport equations on and off between calculations.

1. If your problem involves inflow/outflow, begin by solving the flow without rotation or swirl effects. That is, enable the **Axisymmetric** option instead of the **Axisymmetric Swirl** option in the *General Task Page* (p. 1631), and do not set any rotating boundary conditions. The resulting flow-field data can be used as a starting guess for the full problem.
2. Enable the **Axisymmetric Swirl** option and set all rotating/swirling boundary conditions.
3. Begin the prediction of the rotating/swirling flow by solving only the momentum equation describing the circumferential velocity. This is the **Swirl Velocity** listed in the **Equations** list in the *Equations Dialog Box* (p. 1904). Let the rotation "diffuse" throughout the flow field, based on your boundary condition inputs. In a turbulent flow simulation, you may also want to leave the turbulence equations active during this step. This step will establish the field of rotation throughout the domain.
4. Turn off the momentum equations describing the circumferential motion (**Swirl Velocity**). Leaving the velocity in the circumferential direction fixed, solve the momentum and continuity (pressure) equations (**Flow** in the **Equations** list in the *Equations Dialog Box* (p. 1904)) in the other coordinate directions. This step will establish the axial and radial flows that are a result of the rotation in the field. Again, if your problem involves turbulent flow, you should leave the turbulence equations active during this calculation.
5. Turn on all of the equations simultaneously to obtain a fully coupled solution. Note the under-relaxation controls suggested above.

In addition to the steps above, you may want to simplify your calculation by solving isothermal flow before adding heat transfer or by solving laminar flow before adding a turbulence model. These two methods can be used for any of the solvers (i.e., pressure-based or density-based).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed

Because the rotation or swirl defined by the boundary conditions can lead to large complex forces in the flow, your ANSYS FLUENT calculations will be less stable as the speed of rotation or degree of swirl increases. Hence, one of the most effective controls you can apply to the solution is to solve your rotating flow problem starting with a low rotational speed or swirl velocity and then slowly increase the magnitude up to the desired level. The procedure for accomplishing this is as follows:

1. Set up the problem using a low rotational speed or swirl velocity in your inputs for boundary conditions. The rotation or swirl in this first attempt might be selected as 10% of the actual operating conditions.
2. Solve the problem at these conditions, perhaps using the step-by-step solution strategy outlined above.
3. Save this initial solution data.
4. Modify your inputs (boundary conditions). Increase the speed of rotation, perhaps doubling it.
5. Restart the calculation using the solution data saved in step 3 as the initial solution for the new calculation. Save the new data.
6. Continue to increment the speed of rotation, following steps 4 and 5, until you reach the desired operating condition.

9.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows

Reporting of results for axisymmetric swirling flows is the same as for other flows. The following additional variables are available for postprocessing when axisymmetric swirl is active:

- **Swirl Velocity** (in the **Velocity...** category)
- **Swirl-Wall Shear Stress** (in the **Wall Fluxes...** category)

9.4. Compressible Flows

Compressibility effects are encountered in gas flows at high velocity and/or in which there are large pressure variations. When the flow velocity approaches or exceeds the speed of sound of the gas or when the pressure change in the system ($\Delta p/p$) is large, the variation of the gas density with pressure has a significant impact on the flow velocity, pressure, and temperature. Compressible flows create a unique set of flow physics for which you must be aware of the special input requirements and solution techniques described in this section. *Figure 9.13* (p. 504) and *Figure 9.14* (p. 504) show examples of compressible flows computed using ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.13 Transonic Flow in a Converging-Diverging Nozzle

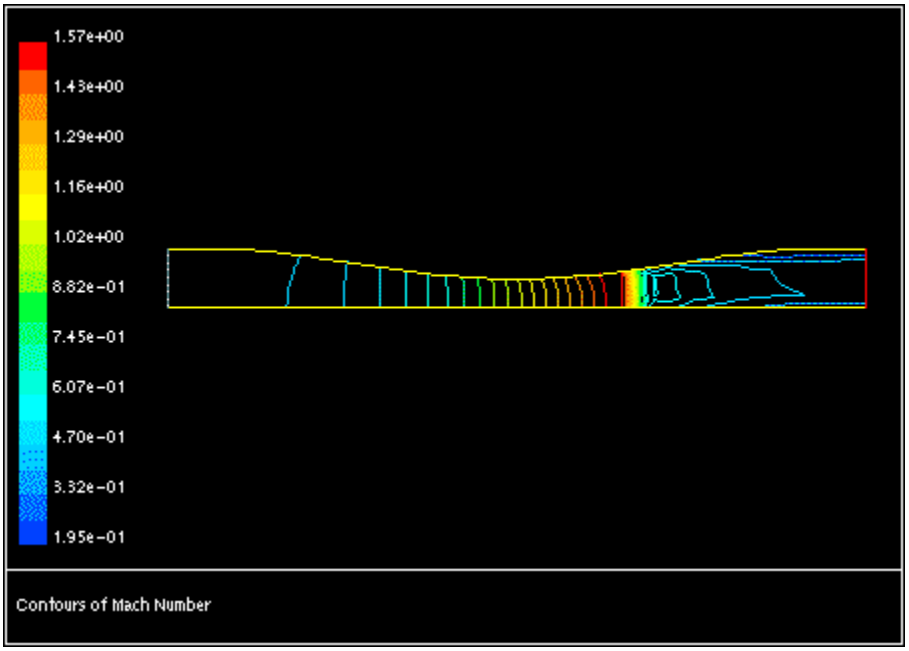
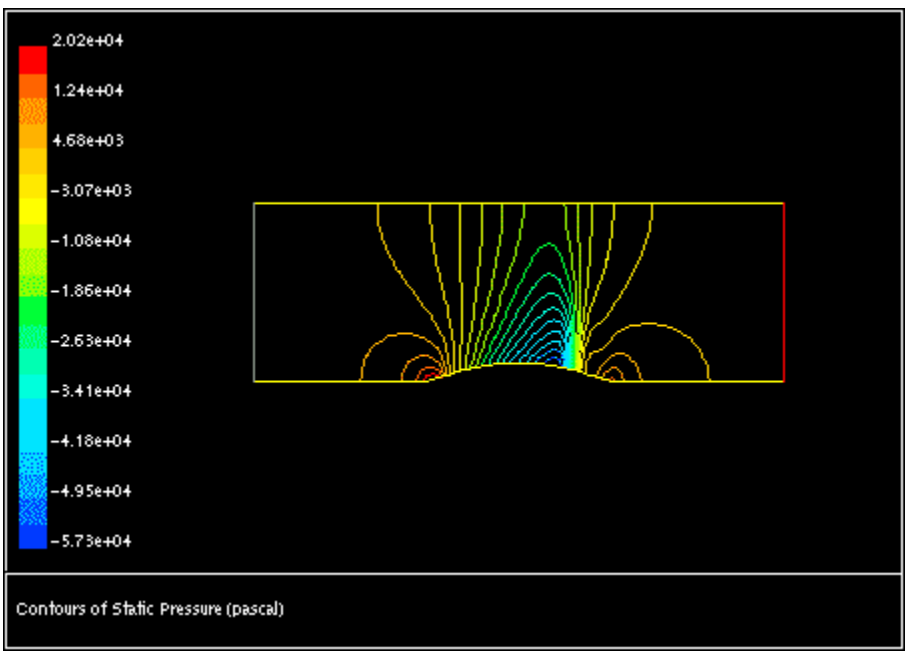


Figure 9.14 Mach 0.675 Flow Over a Bump in a 2D Channel



Information about compressible flows is provided in the following subsections:

9.4.1. When to Use the Compressible Flow Model
9.4.2. Physics of Compressible Flows
9.4.3. Modeling Inputs for Compressible Flows
9.4.4. Floating Operating Pressure
9.4.5. Solution Strategies for Compressible Flows
9.4.6. Reporting of Results for Compressible Flows

For more information about the theoretical background of compressible flows, see Compressible Flows in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.1. When to Use the Compressible Flow Model

Compressible flows can be characterized by the value of the Mach number:

$$M \equiv u/c \tag{9–15}$$

Here, c is the speed of sound in the gas:

$$c = \sqrt{\gamma R T} \tag{9–16}$$

and γ is the ratio of specific heats $\left(c_p/c_v\right)$.

When the Mach number is less than 1.0, the flow is termed subsonic. At Mach numbers much less than 1.0 ($M 0.1$ or so), compressibility effects are negligible and the variation of the gas density with pressure can safely be ignored in your flow modeling. As the Mach number approaches 1.0 (which is referred to as the transonic flow regime), compressibility effects become important. When the Mach number exceeds 1.0, the flow is termed supersonic, and may contain shocks and expansion fans which can impact the flow pattern significantly. ANSYS FLUENT provides a wide range of compressible flow modeling capabilities for subsonic, transonic, and supersonic flows.

9.4.2. Physics of Compressible Flows

Compressible flows are typically characterized by the total pressure p_0 and total temperature T_0 of the flow. For an ideal gas, these quantities can be related to the static pressure and temperature by the following:

$$\frac{p_0}{p} = exp\left(\frac{\int_T^{T_0} \frac{C_p}{T} dT}{R}\right) \tag{9–17}$$

For constant C_p, *Equation 9–17* (p. 505) reduces to

$$\begin{aligned} \frac{p_0}{p} &= \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/(\gamma-1)} \\ \frac{T_0}{T} &= 1 + \frac{\gamma - 1}{2} M^2 \end{aligned} \tag{9–18}$$

These relationships describe the variation of the static pressure and temperature in the flow as the velocity (Mach number) changes under isentropic conditions. For example, given a pressure ratio from inlet to exit (total to static), *Equation 9–18* (p. 505) can be used to estimate the exit Mach number which would exist in a one-dimensional isentropic flow. For air, *Equation 9–18* (p. 505) p/p_0, of 0.5283. This choked flow condition will be established at the point of minimum flow area (e.g., in the throat of a nozzle). In the subsequent area expansion the flow may either accelerate to a supersonic flow in which the pressure will continue to drop, or return to subsonic flow conditions, decelerating with a pressure rise. If a supersonic flow is exposed to an imposed pressure increase, a shock will occur, with a sudden pressure rise and deceleration accomplished across the shock.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.2.1. Basic Equations for Compressible Flows

Compressible flows are described by the standard continuity and momentum equations solved by ANSYS FLUENT, and you do not need to activate any special physical models (other than the compressible treatment of density as detailed below). The energy equation solved by ANSYS FLUENT correctly incorporates the coupling between the flow velocity and the static temperature, and should be activated whenever you are solving a compressible flow. In addition, if you are using the pressure-based solver, you should activate the viscous dissipation terms in Equation 5–1 in the Theory Guide, which become important in high-Mach-number flows.

9.4.2.2. The Compressible Form of the Gas Law

For compressible flows, the ideal gas law is written in the following form:

$$\rho = \frac{p_{op} + p}{\frac{R}{M_w} T} \tag{9–19}$$

where p_{op} is the operating pressure defined in the *Operating Conditions Dialog Box* (p. 1806), p is the local static pressure relative to the operating pressure, R is the universal gas constant, and M_w is the molecular weight. The temperature, T, will be computed from the energy equation.

9.4.3. Modeling Inputs for Compressible Flows

To set up a compressible flow in ANSYS FLUENT, you will need to follow the steps listed below. (Only those steps relevant specifically to the setup of compressible flows are listed here. You will need to set up the rest of the problem as usual.)

1. Set the **Operating Pressure** in the *Operating Conditions Dialog Box* (p. 1806).

   Boundary Conditions → **Operating Conditions...**

   (You can think of p_{op} as the absolute static pressure at a point in the flow where you will define the gauge pressure p to be zero. See *Operating Pressure* (p. 449) for guidelines on setting the operating pressure. For time-dependent compressible flows, you may want to specify a floating operating pressure instead of a constant operating pressure. See *Floating Operating Pressure* (p. 507) for details.)

2. Activate solution of the energy equation in the *Energy Dialog Box* (p. 1645).

   Models → **Energy** → **Edit...**

3. (Pressure-based solver only) If you are modeling turbulent flow, activate the optional viscous dissipation terms in the energy equation by turning on **Viscous Heating** in the *Viscous Model Dialog Box* (p. 1645). Note that these terms can be important in high-speed flows.

   Models → **Viscous** → **Edit...**

   This step is not necessary if you are using one of the density-based solvers, because the density-based solvers always include the viscous dissipation terms in the energy equation.

4. Set the following items in the *Create/Edit Materials Dialog Box* (p. 1736):

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials → **Create/Edit...**

a. Select **ideal-gas** in the drop-down list next to **Density**.

b. Define all relevant properties (specific heat, molecular weight, thermal conductivity, etc.).

5. Set cell zone conditions and boundary conditions (using the *Boundary Conditions Task Page* (p. 1812) and *Cell Zone Conditions Task Page* (p. 1794)), being sure to choose a well-posed cell zone or boundary condition combination that is appropriate for the flow regime. See below for details. Recall that all inputs for pressure (either total pressure or static pressure) must be relative to the operating pressure, and the temperature inputs at inlets should be total (stagnation) temperatures, *not* static temperatures.

Cell Zone Conditions

Boundary Conditions

These inputs should ensure a well-posed compressible flow problem. You will also want to consider special solution parameter settings, as noted in *Solution Strategies for Compressible Flows* (p. 509), before beginning the flow calculation.

9.4.3.1. Boundary Conditions for Compressible Flows

Well-posed inlet and exit boundary conditions for compressible flow are listed below:

- For flow inlets:
  - Pressure inlet: Inlet total temperature and total pressure and, for supersonic inlets, static pressure
  - Mass flow inlet: Inlet mass flow and total temperature
- For flow exits:
  - Pressure outlet: Exit static pressure (ignored if flow is supersonic at the exit. All the information travels downstream in a supersonic region, hence the pressure at the outlet can be computed by directly extrapolating from the adjacent cell center [*33*] (p. 2210). Therefore, it is not meaningful to use the exit static pressure prescribed in the boundary conditions task page, and the exit static pressure is ignored).

It is important to note that your boundary condition inputs for pressure (either total pressure or static pressure) must be in terms of gauge pressure — i.e., pressure relative to the operating pressure defined in the *Operating Conditions Dialog Box* (p. 1806), as described above.

All temperature inputs at inlets should be total (stagnation) temperatures, *not* static temperatures.

9.4.4. Floating Operating Pressure

ANSYS FLUENT provides a "floating operating pressure" option to handle time-dependent compressible flows with a gradual increase in the absolute pressure in the domain. This option is desirable for slow subsonic flows with static pressure build-up, since it efficiently accounts for the slow changing of absolute pressure without using acoustic waves as the transport mechanism for the pressure build-up.

Examples of typical applications include the following:

- combustion or heating of a gas in a closed domain
- pumping of a gas into a closed domain

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.4.1. Limitations

The floating operating pressure option should *not* be used for transonic or incompressible flows. In addition, it cannot be used if your model includes any pressure inlet, pressure outlet, exhaust fan, inlet vent, intake fan, outlet vent, or pressure far field boundaries.

9.4.4.2. Theory

The floating operating pressure option allows ANSYS FLUENT to calculate the pressure rise (or drop) from the integral mass balance, separately from the solution of the pressure correction equation. When this option is activated, the absolute pressure at each iteration can be expressed as

$$p_{abs} = p_{op,float} + p \tag{9–20}$$

where p is the pressure relative to the reference location, which in this case is in the cell with the minimum pressure value. Thus the reference location itself is floating.

$p_{op,float}$ is referred to as the floating operating pressure, and is defined as

$$p_{op,float} = p_{op}^{0} + \Delta p_{op} \tag{9–21}$$

where p_{op}^{0} is the initial operating pressure and Δp_{op} is the pressure rise.

Including the pressure rise Δp_{op} in the floating operating pressure $p_{op,float}$, rather than in the pressure p, helps to prevent roundoff error. If the pressure rise were included in p, the calculation of the pressure gradient for the momentum equation would give an inexact balance due to precision limits for 32-bit real numbers.

9.4.4.3. Enabling Floating Operating Pressure

When time dependence is active, you can turn on the **Floating Operating Pressure** option in the *Operating Conditions Dialog Box* (p. 1806).

Boundary Conditions → **Operating Conditions...**

(Note that the inputs for **Reference Pressure Location** will disappear when you enable **Floating Operating Pressure**, since these inputs are no longer relevant.)

Important

The floating operating pressure option should *not* be used for transonic flows or for incompressible flows. It is meaningful only for slow subsonic flows of ideal gases, when the characteristic time scale is much larger than the sonic time scale.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.4.4. Setting the Initial Value for the Floating Operating Pressure

When the floating operating pressure option is enabled, you will need to specify a value for the **Initial Operating Pressure** in the *Solution Initialization Task Page* (p. 1937).

Solution Initialization

This initial value is stored in the case file with all your other initial values.

9.4.4.5. Storage and Reporting of the Floating Operating Pressure

The current value of the floating operating pressure is stored in the data file. If you visit the *Operating Conditions Dialog Box* (p. 1806) after a number of time steps have been performed, the current value of the **Operating Pressure** will be displayed.

Note that the floating operating pressure will automatically be reset to the initial operating pressure if you reset the data (i.e., start over at the first iteration of the first time step).

9.4.4.6. Monitoring Absolute Pressure

You can monitor the absolute pressure during the calculation using the *Surface Monitor Dialog Box* (p. 1923) (see *Monitoring Surface Integrals* (p. 1283) for details). You can also generate graphical plots or alphanumeric reports of absolute pressure when your solution is complete. The **Absolute Pressure** variable is contained in the **Pressure...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See *Field Function Definitions* (p. 1529) for its definition.

9.4.5. Solution Strategies for Compressible Flows

The difficulties associated with solving compressible flows are a result of the high degree of coupling between the flow velocity, density, pressure, and energy. This coupling may lead to instabilities in the solution process and, therefore, may require special solution techniques in order to obtain a converged solution. In addition, the presence of shocks (discontinuities) in the flow introduces an additional stability problem during the calculation. Solution techniques that may be beneficial in compressible flow calculations include the following:

- (Pressure-based solver only) Initialize the flow to be near stagnation (i.e. velocity small but not zero, pressure to inlet total pressure, temperature to inlet total temperature). Turn off the energy equation for the first 50 iterations. Leave the energy under-relaxation at 1. Set the pressure under-relaxation to 0.4, and the momentum under-relaxation to 0.3. After the solution stabilizes and the energy equation has been turned on, increase the pressure under-relaxation to 0.7.
- Set reasonable limits for the temperature and pressure (in the *Solution Limits Dialog Box* (p. 1904)) to avoid solution divergence, especially at the start of the calculation. If ANSYS FLUENT prints messages about temperature or pressure being limited as the solution nears convergence, the high or low computed values may be physical, and you will need to change the limits to allow these values.
- If required, begin the calculations using a reduced pressure ratio at the boundaries, increasing the pressure ratio gradually in order to reach the final desired operating condition. If the Mach number is low, you can also consider starting the compressible flow calculation from an incompressible flow solution (although the incompressible flow solution can in some cases be a rather poor initial guess for the compressible calculation).
- In some cases, computing an inviscid solution as a starting point may be helpful.

See *Using the Solver* (p. 1207) for details on the procedures used to make these changes to the solution parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.6. Reporting of Results for Compressible Flows

You can display the results of your compressible flow calculations in the same manner that you would use for an incompressible flow. The variables listed below are of particular interest when you model compressible flow:

- **Total Temperature**
- **Total Pressure**
- **Mach Number**

These variables are contained in the variable selection drop-down list that appears in postprocessing dialog boxes. **Total Temperature** is in the **Temperature...** category, **Total Pressure** is in the **Pressure...** category, and **Mach Number** is in the **Velocity...** category. See *Field Function Definitions* (p. 1529) for their definitions.

9.5. Inviscid Flows

Inviscid flow analysis neglect the effect of viscosity on the flow and are appropriate for high-Reynolds-number applications where inertial forces tend to dominate viscous forces. One example for which an inviscid flow calculation is appropriate is an aerodynamic analysis of some high-speed projectile. In a case like this, the pressure forces on the body will dominate the viscous forces. Hence, an inviscid analysis will give you a quick estimate of the primary forces acting on the body. After the body shape has been modified to maximize the lift forces and minimize the drag forces, you can perform a viscous analysis to include the effects of the fluid viscosity and turbulent viscosity on the lift and drag forces.

Another area where inviscid flow analysis are routinely used is to provide a good initial solution for problems involving complicated flow physics and/or complicated flow geometry. In a case like this, the viscous forces are important, but in the early stages of the calculation the viscous terms in the momentum equations will be ignored. Once the calculation has been started and the residuals are decreasing, you can turn on the viscous terms (by enabling laminar or turbulent flow) and continue the solution to convergence. For some very complicated flows, this is the only way to get the calculation started.

Information about inviscid flows is provided in the following subsections:

9.5.1. Setting Up an Inviscid Flow Model
9.5.2. Solution Strategies for Inviscid Flows
9.5.3. Postprocessing for Inviscid Flows

For more information about the theoretical background of inviscid flows, see Inviscid Flows in the Theory Guide.

9.5.1. Setting Up an Inviscid Flow Model

For inviscid flow problems, you will need to perform the following steps during the problem setup procedure. (Only those steps relevant specifically to the setup of inviscid flow are listed here. You will need to set up the rest of the problem as usual.)

1. Activate the calculation of inviscid flow by selecting **Inviscid** in the *Viscous Model Dialog Box* (p. 1645).

   Models → **Viscous** → **Edit...**

2. Set boundary conditions and flow properties.

   Boundary Conditions

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials

3. Solve the problem and examine the results.

9.5.2. Solution Strategies for Inviscid Flows

Since inviscid flow problems will usually involve high-speed flow, you may have to reduce the under-relaxation factors for momentum (if you are using the pressure-based solver) or reduce the Courant number (if you are using the density-based solver), in order to get the solution started. Once the flow is started and the residuals are monotonically decreasing, you can start increasing the under-relaxation factors or Courant number back up to the default values.

Modifications to the under-relaxation factors and the Courant number can be made in the *Solution Controls Task Page* (p. 1902).

Solution Controls

The solution strategies for compressible flows apply also to inviscid flows. See *Solution Strategies for Compressible Flows* (p. 509) for details.

9.5.3. Postprocessing for Inviscid Flows

If you are interested in the lift and drag forces acting on your model, you can use the *Force Reports Dialog Box* (p. 2029) to compute them.

Reports → **Forces** → **Set Up...**

See *Forces on Boundaries* (p. 1515) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Modeling Flows with Moving Reference Frames

This chapter provides details about the moving reference frame capabilities in ANSYS FLUENT.

The information in this chapter is divided into the following sections:

10.1. Introduction
10.2. Flow in Single Moving Reference Frames (SRF)
10.3. Flow in Multiple Moving Reference Frames

10.1. Introduction

ANSYS FLUENT solves the equations of fluid flow and heat transfer, by default, in a stationary (or inertial) reference frame. However, there are many problems where it is advantageous to solve the equations in a moving (or non-inertial) reference frame. Such problems typically involve moving parts (such as rotating blades, impellers, and similar types of moving surfaces), and it is the flow around these moving parts that is of interest. In most cases, the moving parts render the problem unsteady when viewed from the stationary frame. With a moving reference frame, however, the flow around the moving part can (with certain restrictions) be modeled as a steady-state problem with respect to the moving frame.

ANSYS FLUENT's moving reference frame modeling capability allows you to model problems involving moving parts by allowing you to activate moving reference frames in selected cell zones. When a moving reference frame is activated, the equations of motion are modified to incorporate the additional acceleration terms which occur due to the transformation from the stationary to the moving reference frame. By solving these equations in a steady-state manner, the flow around the moving parts can be modeled.

For many problems, it may be possible to refer the entire computational domain to a single moving reference frame. This is known as the single reference frame (or SRF) approach. The use of the SRF approach is possible; provided the geometry meets certain requirements. For more complex geometries, it may not be possible to use a single reference frame. In such cases, you must break up the problem into multiple cell zones, with well-defined interfaces between the zones. The manner in which the interfaces are treated leads to two approximate, steady-state modeling methods for this class of problem: the multiple reference frame (or MRF) approach, and the mixing plane approach. These approaches will be discussed in *The Multiple Reference Frame Model* (p. 522) and *The Mixing Plane Model* (p. 523). If unsteady interaction between the stationary and moving parts is important, you can employ the sliding mesh approach to capture the transient behavior of the flow. The sliding meshing model will be discussed in *Modeling Flows Using Sliding and Dynamic Meshes* (p. 535).

The principal reason for employing a moving reference frame is to render a problem which is unsteady in the stationary (inertial) frame steady with respect to the moving frame. For a steadily moving frame (e.g., the frame speed is constant), it is possible to transform the equations of fluid motion to the moving frame such that steady-state solutions are possible. By default, ANSYS FLUENT permits the activation of a moving reference frame with a steady speed. If the speed is not constant, the transformed equations will contain additional terms (see Relative Velocity Formulation in the Theory Guide). It should also be noted that you can run an unsteady simulation in a moving reference frame with constant speed. This would be necessary if you wanted to simulate, for example, vortex shedding from a rotating fan blade. The unsteadiness in this case is due to a natural fluid instability (vortex generation) rather than induced from interaction with a stationary component.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about the equations for moving reference frames, see Equations for a Moving Reference Frame in the Theory Guide.

Figure 10.1 Single Component (Blower Wheel Blade Passage)

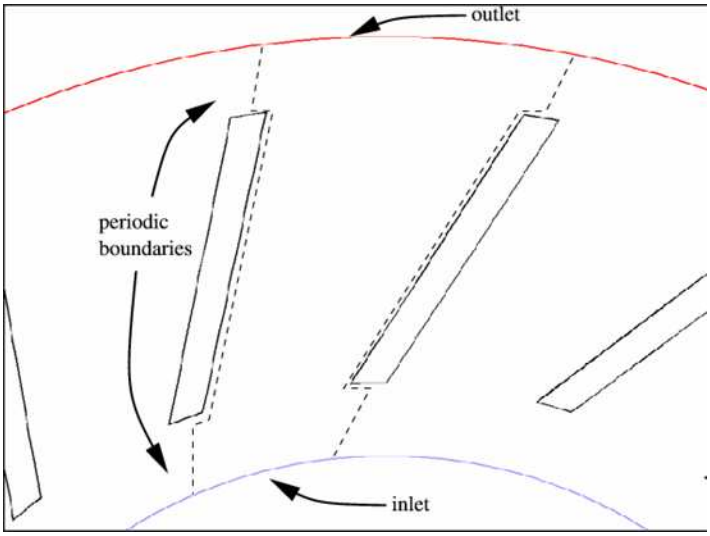
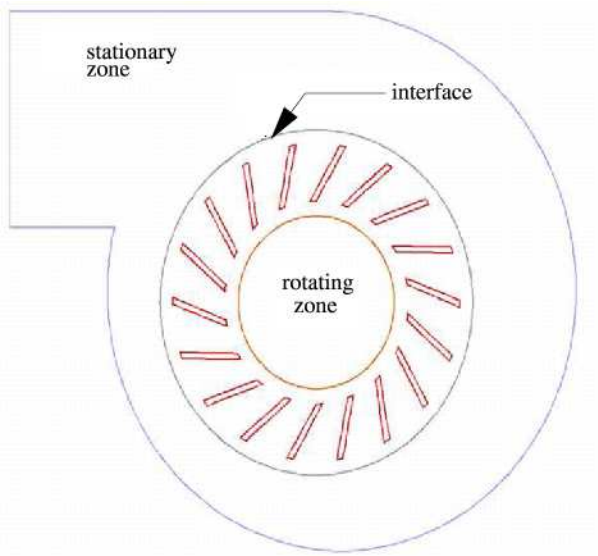


Figure 10.2 Multiple Component (Blower Wheel and Casing)



10.2. Flow in Single Moving Reference Frames (SRF)

Many problems permit the entire computational domain to be referred to a single moving reference frame (hence the name SRF modeling). In such cases, the equations given in Equations for a Moving Reference Frame are solved in all fluid cell zones. Steady-state solutions are possible in SRF models provided suitable boundary conditions are prescribed. In particular, wall boundaries must adhere to the following requirements:

- Any walls which are moving with the reference frame can assume any shape. An example would be the blade surfaces associated with a pump impeller. The no slip condition is defined in the relative frame such that the relative velocity is zero on the moving walls.
- For a rotating problem, you can define walls that are non-moving with respect to the stationary coordinate system, but these walls must be surfaces of revolution about the axis of rotation. Here the no slip

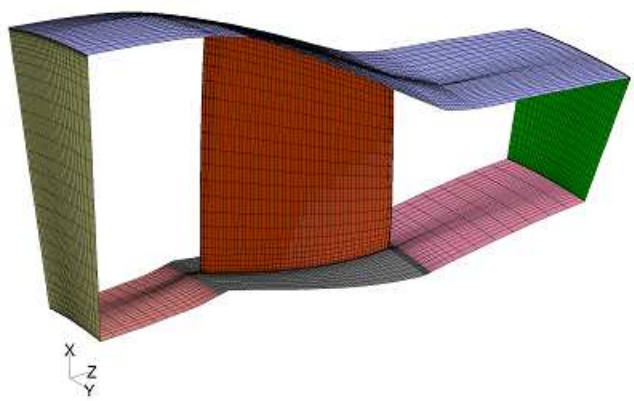
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

condition is defined such that the absolute velocity is zero on the walls. An example of this type of boundary would be a cylindrical wind tunnel wall which surrounds a rotating propeller.

Rotationally periodic boundaries may also be used, but the surface must be periodic about the axis of rotation. As an example, it is very common to model flow through a blade row of a turbomachine by assuming the flow to be rotationally periodic and using a periodic domain about a single blade. This permits good resolution of the flow around the blade without the expense of modeling all blades in the blade row (see *Figure 10.3* (p. 515)).

Flow boundary conditions in ANSYS FLUENT (inlets and outlets) can, in most cases, be prescribed in either the stationary or moving frames. For example, for a velocity inlet, one can specify either the relative velocity or absolute velocity, depending on which is more convenient. For additional information on these and other boundary conditions, see *Setting Up a Single Moving Reference Frame Problem* (p. 516) and *Cell Zone and Boundary Conditions* (p. 205).

Figure 10.3 Single Blade Model with Rotationally Periodic Boundaries



10.2.1. Mesh Setup for a Single Moving Reference Frame

It is important to remember the following coordinate-system constraints when you are setting up a problem involving a moving reference frame for a rotating problem:

- For 2D problems, the axis of rotation must be parallel to the z axis.
- For 2D axisymmetric problems, the axis of rotation must be the x axis.
- For 3D geometries, you should generate the mesh with a specific rotational axis in mind for the rotating cell zone. Usually it is convenient to use the x, y, or z axis, but ANSYS FLUENT can accommodate arbitrary rotational axes.

With 3D rotating problems, it is also important to note that if you wish to include walls which have zero velocity in the stationary frame, these walls must be a surface of revolution with respect to the axis of rotation. If the stationary walls are not surfaces of revolution, you must encapsulate the rotating parts with interface boundaries, thereby breaking your model up into multiple zones, and use either the MRF or mixing plane models for steady state solutions (see *The Multiple Reference Frame Model* (p. 522) and *The Mixing Plane Model* (p. 523)), or the sliding mesh model for unsteady interaction (see *Modeling Flows Using Sliding and Dynamic Meshes* (p. 535)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.2.2. Setting Up a Single Moving Reference Frame Problem

To model a problem involving a single moving reference frame, follow the steps outlined below.

1. Select the **Velocity Formulation** to be used when solving: either **Relative** or **Absolute**. (See *Choosing the Relative or Absolute Velocity Formulation* (p. 518) for details.)

   General

   (Note that this step is irrelevant if you are using one of the density-based solvers; these solvers always use an absolute velocity formulation.)

2. For each cell zone in the domain, specify the translational velocity of the reference frame and/or the angular velocity (ω) of the reference frame and the axis about which it rotates.

   Cell Zone Conditions

   a. In the **Fluid** or **Solid** dialog box, specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the frame motion to define the axis of rotation.

   b. Also in the **Fluid** (*Figure 10.4* (p. 517)) or **Solid** dialog box, enable the **Frame Motion** option and then set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in *Defining Transient Cell Zone and Boundary Conditions* (p. 376), or a UDF macro, described in DEFINE_TRANSIENT_PROFILE. Specifying the individual velocities as either a profile or a UDF allows you to specify a specific input of the frame motion individually. However, you can also specify the frame motion inputs via a single user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

   Note

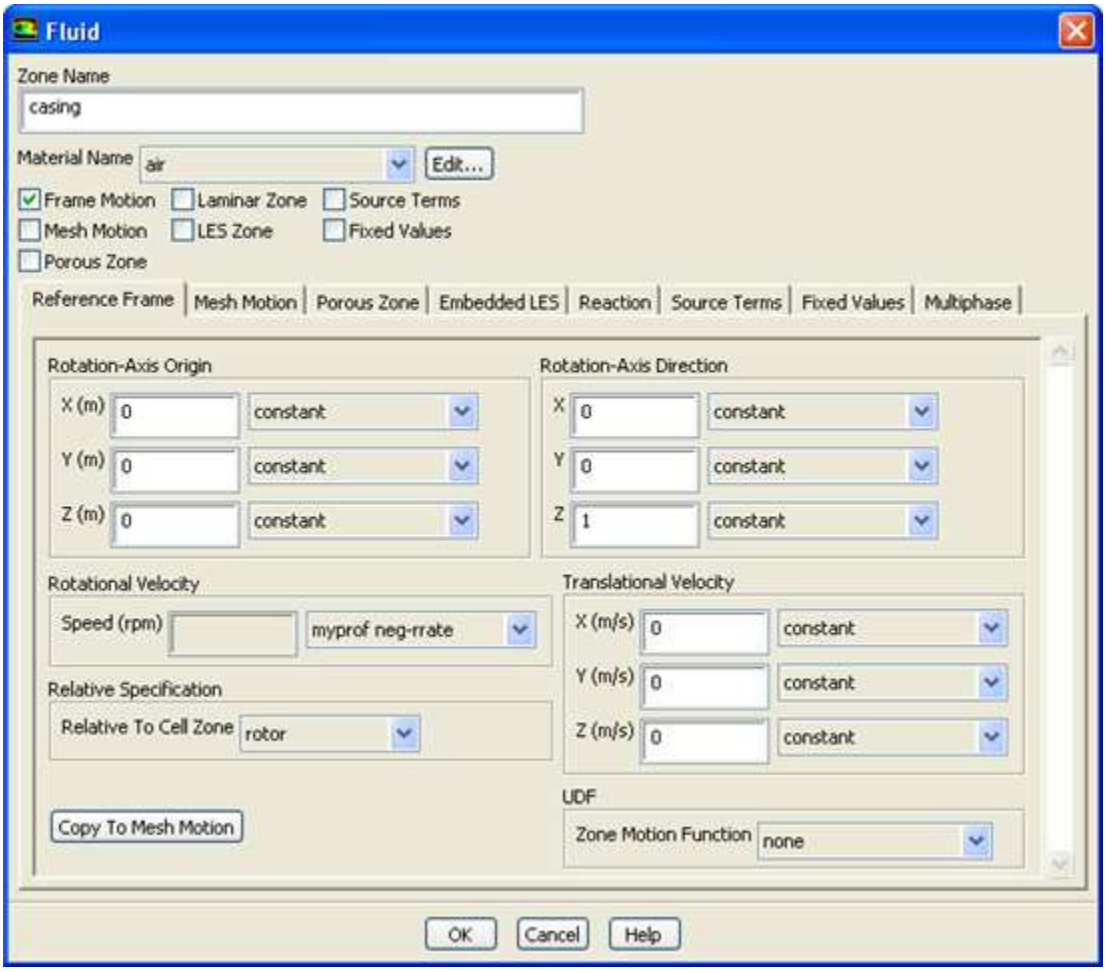
   If you decide to hook a UDF, you will no longer have access to the rotation axis origin and direction, or the velocities.

   Details about these inputs are presented in *Inputs for Fluid Zones* (p. 215) and in *Inputs for Solid Zones* (p. 218). Details about the zone motion UDF can be found in DEFINE_ZONE_MOTION in the UDF Manual.

   c. If you need to switch between the moving reference frame and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.4 The Fluid Dialog Box Displaying Frame Motion Inputs



Important

For solid zones, you only need to activate the **Frame Motion** option if you intend to include the convective terms in the energy equation for the solid (Equation 5–11 in the Theory Guide). Normally, this is not required if you wish to do a conjugate heat transfer problem where the solid and fluid zones are moving together.

3. Define the velocity boundary conditions at walls. You can choose to define either an absolute velocity or a velocity relative to the moving reference frame (i.e., relative to the velocity of the adjacent cell zone specified in step 2).

   If the wall is moving at the speed of the moving frame (and hence stationary in the moving frame), it is convenient to specify a relative angular velocity of zero. Likewise, a wall that is stationary in the non-moving frame of reference should be given a velocity of zero in the absolute reference frame. Specifying the wall velocities in this manner obviates the need to modify these inputs later if a change is made in the velocity of the fluid zone.

   Details about these inputs are presented in *Velocity Conditions for Moving Walls* (p. 302).

4. Define the boundary conditions at the inlets, as described in *Boundary Conditions* (p. 249). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

adjacent cell zone (specified in step 2). Likewise, the total pressure and flow direction can be prescribed in absolute or relative frames for pressure inlets.

Details about these inputs are presented in *Defining the Flow Direction* (p. 259) and *Defining the Velocity* (p. 266).

10.2.2.1. Choosing the Relative or Absolute Velocity Formulation

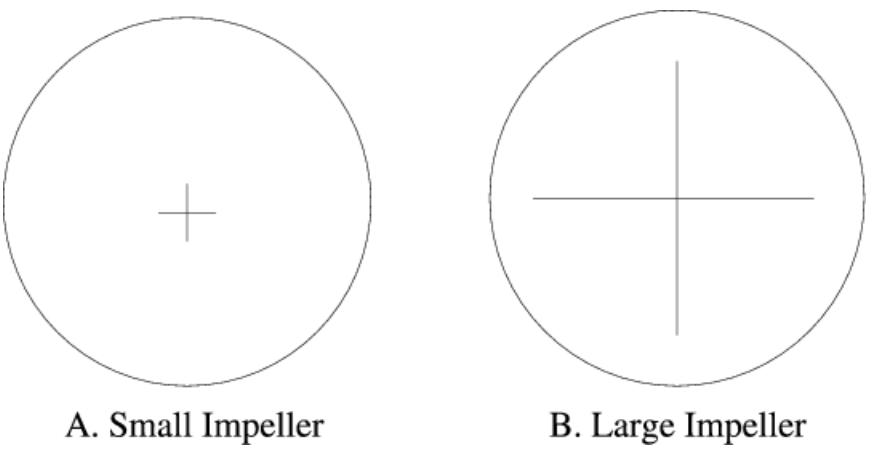
It is recommended that you use the velocity formulation that will result in most of the flow domain having the smallest velocities in that frame, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution.

The absolute velocity formulation is preferred in applications where the flow in most of the domain is not moving (e.g., a fan in a large room). The relative velocity formulation is appropriate when most of the fluid in the domain is moving, as in the case of a large impeller in a mixing tank.

10.2.2.1.1. Example

A problem with stationary outer walls and a rotating impeller can be solved in a single reference frame. The example is illustrated in *Figure 10.5* (p. 518).

Figure 10.5 Geometry with the Rotating Impeller



In case A, it is expected that only the flow near the impeller would be rotating and that much of the flow away from the impeller would have a low velocity magnitude in the absolute frame. Therefore, solving using the absolute velocity formulation is recommended. In case B, most of the flow is expected to be rotating with a velocity close to that of the impeller. Hence, the relative velocity formulation is appropriate.

In a situation between case A and case B, either of the formulations may be used.

Important

- If the velocity formulation is switched during the solution process, ANSYS FLUENT will not transform the current solution to the other frame, which can lead to large jumps in residuals. If changing the frame is necessary, it is recommended that you first reinitialize, and then solve.
- When one of the density-based solution algorithms is used, the absolute formulation is always used; the relative velocity formulation is not available in the density-based solvers.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For velocity inlets, pressure inlets, mass flow inlets, and walls, you may specify velocity in either the absolute or the relative frame, regardless of whether the absolute or relative velocity formulation is used in the computation.

For pressure outlets, the specified static pressure is independent of frame. However, when there is backflow at a pressure outlet, the specified static pressure is used as the total pressure. For calculations using the absolute velocity formulation, the specified static pressure is used as the total pressure in the absolute frame; for the relative velocity formulation, the specified static pressure is assumed to be the total pressure in the relative frame. As for the flow direction, ANSYS FLUENT assumes the absolute velocity to be normal to the pressure outlet for the absolute velocity formulation; for the relative velocity formulation, it is the relative velocity that is assumed to be normal to the pressure outlet.

10.2.3. Solution Strategies for a Single Moving Reference Frame

The difficulties associated with solving flows in moving reference frames are similar to those discussed in *Solution Strategies for Axisymmetric Swirling Flows* (p. 501). The primary issue you must confront is the high degree of coupling between the momentum equations when the influence of the rotational terms is large. A high degree of rotation introduces a large radial pressure gradient which drives the flow in the axial and radial directions, thereby setting up a distribution of the swirl or rotation in the field. This coupling may lead to instabilities in the solution process, and hence require special solution techniques to obtain a converged solution. Some techniques that may be beneficial include the following:

- (Pressure-based solver only) Consider switching the frame in which velocities are solved by changing the velocity formulation setting in the *General Task Page* (p. 1631). (See *Choosing the Relative or Absolute Velocity Formulation* (p. 518) for details.)
- (Pressure-based segregated solver only) Use the PRESTO! scheme (enabled in the *Solution Methods Task Page* (p. 1900)), which is well-suited for the steep pressure gradients involved in rotating flows.
- Ensure that the mesh is sufficiently refined to resolve large gradients in pressure and swirl velocity.
- (Pressure-based, segregated solver only) Reduce the under-relaxation factors for the velocities, perhaps to 0.3–0.5 or lower, if necessary.
- Begin the calculations using a low rotational speed, increasing the rotational speed gradually in order to reach the final desired operating condition.

See *Using the Solver* (p. 1207) for details on the procedures used to make these changes to the solution parameters.

10.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability

Because the rotation of the reference frame and the rotation defined via boundary conditions can lead to large complex forces in the flow, your ANSYS FLUENT calculations may be less stable as the speed of rotation (and hence the magnitude of these forces) increases. One of the most effective controls you can exert on the solution is to start with a low rotational speed and then slowly increase the rotation up to the desired level. The procedure you use to accomplish this is as follows:

1. Set up the problem using a low rotational speed in your inputs for boundary conditions and for the angular velocity of the reference frame. The rotational speed in this first attempt might be selected as 10% of the actual operating condition.
2. Solve the problem at these conditions.
3. Save this initial solution data.
4. Modify your inputs (i.e., boundary conditions and angular velocity of the reference frame). Increase the speed of rotation, perhaps doubling it.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

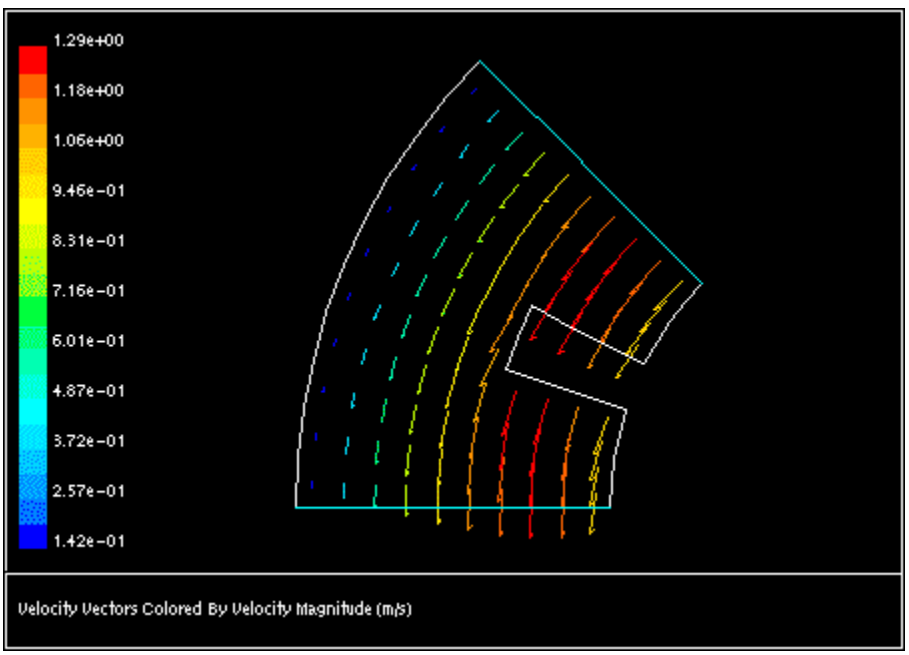
5. Restart or continue the calculation using the solution data saved in Step 3 as the initial guess for the new calculation. Save the new data.
6. Continue to increment the rotational speed, following Steps 4 and 5, until you reach the desired operating condition.

10.2.4. Postprocessing for a Single Moving Reference Frame

When you solve a problem in a moving reference frame, you can plot or report both absolute and relative velocities. For all velocity parameters (e.g., **Velocity Magnitude** and **Mach Number**), corresponding relative values will be available for postprocessing (e.g., **Relative Velocity Magnitude** and **Relative Mach Number**). These variables are contained in the **Velocity...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Relative values are also available for postprocessing of total pressure, total temperature, and any other parameters that include a dynamic contribution dependent on the reference frame (e.g., **Relative Total Pressure**, **Relative Total Temperature**, **Rothalpy**).

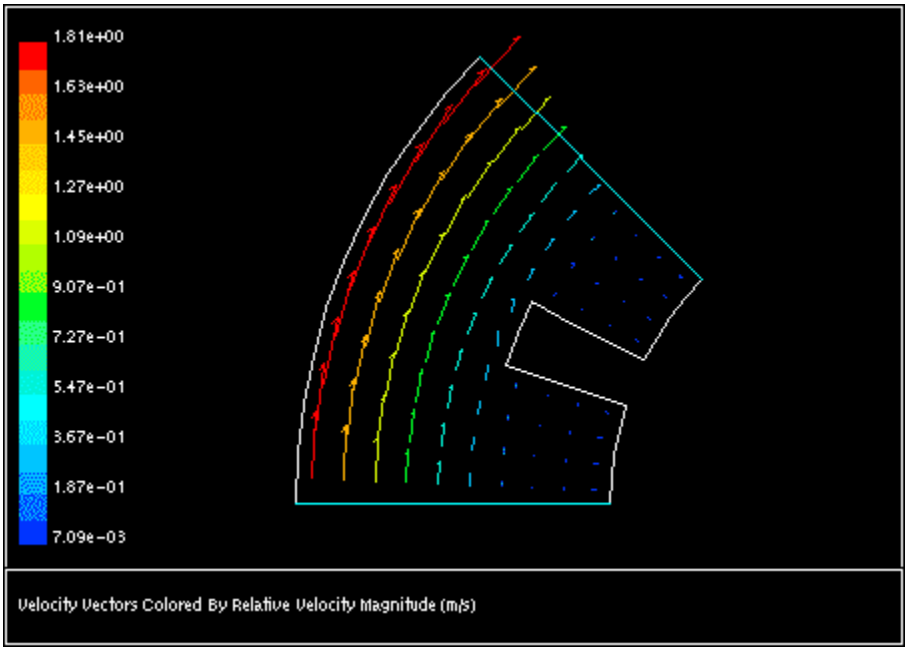
When plotting velocity vectors, you can choose to plot vectors in the absolute frame (the default), or you can select **Relative Velocity** in the **Vectors of** drop-down list in the *Vectors Dialog Box* (p. 1970) to plot vectors in the moving frame. If you plot relative velocity vectors, you might want to color the vectors by relative velocity magnitude (by choosing **Relative Velocity Magnitude** in the **Color by** list); by default they will be colored by absolute velocity magnitude. *Figure 10.6* (p. 520) and *Figure 10.7* (p. 521) show absolute and relative velocity vectors in a moving domain with a stationary outer wall.

Figure 10.6 Absolute Velocity Vectors



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.7 Relative Velocity Vectors



10.3. Flow in Multiple Moving Reference Frames

Many problems involve multiple moving parts or contain stationary surfaces which are not surfaces of revolution (and therefore cannot be used with the Single Reference Frame modeling approach). For these problems, you must break up the model into multiple fluid/solid cell zones, with interface boundaries separating the zones. Zones which contain the moving components can then be solved using the moving reference frame equations (Equations for a Moving Reference Frame in the Theory Guide), whereas stationary zones can be solved with the stationary frame equations. The manner in which the equations are treated at the interface lead to two approaches which are supported in ANSYS FLUENT:

- Multiple Moving Reference Frames
  - Multiple Reference Frame Model (MRF)
  - Mixing Plane Model (MPM)
- Sliding Mesh Model (SMM)

Both the MRF and mixing plane approaches are steady-state approximations, and differ primarily in the manner in which conditions at the interfaces are treated. These approaches will be discussed in the sections below. The sliding mesh model approach is, on the other hand, inherently unsteady due to the motion of the mesh with time. This approach is discussed in *Modeling Flows Using Sliding and Dynamic Meshes* (p. 535).

For additional information, please see the following sections:

10.3.1. The Multiple Reference Frame Model
10.3.2. The Mixing Plane Model
10.3.3. Mesh Setup for a Multiple Moving Reference Frame
10.3.4. Setting Up a Multiple Moving Reference Frame Problem
10.3.5. Solution Strategies for MRF and Mixing Plane Problems
10.3.6. Postprocessing for MRF and Mixing Plane Problems

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.1. The Multiple Reference Frame Model

10.3.1.1. Overview

The MRF model [*48*] (p. 2211) is, perhaps, the simplest of the two approaches for multiple zones. It is a steady-state approximation in which individual cell zones can be assigned different rotational and/or translational speeds. The flow in each moving cell zone is solved using the moving reference frame equations (see *Introduction* (p. 513)). If the zone is stationary ($\omega = 0$), the equations reduce to their stationary forms. At the interfaces between cell zones, a local reference frame transformation is performed to enable flow variables in one zone to be used to calculate fluxes at the boundary of the adjacent zone. For more information about the MRF interface formulation,

It should be noted that the MRF approach does not account for the relative motion of a moving zone with respect to adjacent zones (which may be moving or stationary); the mesh remains fixed for the computation. This is analogous to freezing the motion of the moving part in a specific position and observing the instantaneous flowfield with the rotor in that position. Hence, the MRF is often referred to as the "frozen rotor approach."

While the MRF approach is clearly an approximation, it can provide a reasonable model of the flow for many applications. For example, the MRF model can be used for turbomachinery applications in which rotor-stator interaction is relatively weak, and the flow is relatively uncomplicated at the interface between the moving and stationary zones. In mixing tanks, for example, since the impeller-ba e interactions are relatively weak, large-scale transient effects are not present and the MRF model can be used.

Another potential use of the MRF model is to compute a flow field that can be used as an initial condition for a transient sliding mesh calculation. This eliminates the need for a startup calculation. The multiple reference frame model should not be used, however, if it is necessary to actually simulate the transients that may occur in strong rotor-stator interactions, the sliding mesh model alone should be used (see *Modeling Flows Using Sliding and Dynamic Meshes* (p. 535).

For more information about and examples of multiple moving reference frames, see The Multiple Reference Frame Model in the Theory Guide.

10.3.1.2. Limitations

The following limitations exist when using the MRF approach:

- The interfaces separating a moving region from adjacent regions must be oriented such that the component of the frame velocity normal to the boundary is zero. This means that for a translationally moving frame, the moving zone's boundaries must be parallel to the translational velocity vector. For rotating problems, the interfaces must be surfaces of revolution about the axis of rotation defined for the fluid zone. For the example shown Figure 2.4: "Geometry with One Rotating Impeller" (in the Theory Guide), this requires the dashed boundary to be circular (not square or any other shape).
- Strictly speaking, the use of multiple reference frames is meaningful only for steady flow. However, ANSYS FLUENT will allow you to solve an unsteady flow when multiple reference frames are being used. In this case, unsteady terms (as described in Temporal Discretization in the Theory Guide) are added to all the governing transport equations. You should carefully consider whether this will yield meaningful results for your application, because, for unsteady flows, a sliding mesh calculation will generally yield more meaningful results than an MRF calculation.
- Particle trajectories and pathlines drawn by ANSYS FLUENT use the velocity relative to the cell zone motion. For massless particles, the resulting pathlines follow the streamlines based on relative velocity.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For particles with mass, however, the particle tracks displayed are meaningless. Similarly, coupled discrete-phase calculations are meaningless.

An alternative approach for particle tracking and coupled discrete-phase calculations with multiple reference frames is to track particles based on absolute velocity instead of relative velocity. To make this change, use the `define/models/dpm/options/track-in-absolute-frame` text command. Note, that the results may strongly depend on the location of walls inside the multiple reference frame. The particle injection velocities (specified in the *Set Injection Properties Dialog Box* (p. 2097)) are defined relative to the frame of reference in which the particles are tracked. By default, the injection velocities are specified relative to the local reference frame. If you enable the `track-in-absolute-frame` option, the injection velocities are specified relative to the absolute frame.

- You cannot accurately model axisymmetric swirl in the presence of multiple reference frames using the relative velocity formulation. This is because the current implementation does not apply the transformation used in Equation 2–16 (in the Theory Guide) to the swirl velocity derivatives. For this situation, the absolute velocity formulation should be used.
- Translational and rotational velocities are assumed to be constant (time varying ω, v_t are not allowed).
- The relative velocity formulation cannot be used in combination with the MRF and mixture models. (For details, see Mixture Model Theory in the Theory Guide). For such cases, use the absolute velocity formulation instead.
- You must not have a single interface between reference frames where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (i.e., the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169) for the steps involved in setting up a coupled interface.

Important

You can switch from the MRF model to the sliding mesh model for a more robust and accurate solution, by using the `mesh/modify-zones/mrf-to-sliding-mesh` text command. See *Using Sliding Meshes* (p. 540) for details on how to make this change in the fluid's boundary conditions. Currently, this switch is not possible when running in parallel or for non-conformal interfaces.

10.3.2. The Mixing Plane Model

The mixing plane model in ANSYS FLUENT provides an alternative to the multiple reference frame and sliding mesh models for simulating flow through domains with one or more regions in relative motion. This section provides a brief overview of the model and a list of its limitations.

10.3.2.1. Overview

As discussed in *The Multiple Reference Frame Model* (p. 522), the MRF model is applicable when the flow at the interface between adjacent moving/stationary zones is nearly uniform ("mixed out"). If the flow at this interface is not uniform, the MRF model may not provide a physically meaningful solution. The sliding mesh model (see *Modeling Flows Using Sliding and Dynamic Meshes* (p. 535)) may be appropriate for such cases, but in many situations it is not practical to employ a sliding mesh. For example, in a multistage turbomachine, if the number of blades is different for each blade row, a large number of blade passages is required in order to maintain circumferential periodicity. Moreover, sliding mesh calculations are necessarily unsteady, and thus require significantly more computation to achieve a final, time-periodic solution. For situations where using the sliding mesh model is not feasible, the mixing plane model can be a cost-effective alternative.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the mixing plane approach, each fluid zone is treated as a steady-state problem. Flow-field data from adjacent zones are passed as boundary conditions that are spatially averaged or "mixed" at the mixing plane interface. This mixing removes any unsteadiness that would arise due to circumferential variations in the passage-to-passage flow field (e.g., wakes, shock waves, separated flow), thus yielding a steady-state result. Despite the simplifications inherent in the mixing plane model, the resulting solutions can provide reasonable approximations of the time-averaged flow field.

10.3.2.2. Limitations

Note the following limitations of the mixing plane model:

- The LES turbulence model cannot be used with the mixing plane model.
- The models for species transport and combustion cannot be used with the mixing plane model.
- The VOF multiphase model cannot be used with the mixing plane model.
- The discrete phase model cannot be used with the mixing plane model for coupled flows. Non-coupled computations can be done, but you should note that the particles leave the domain of the mixing plane.

You can find more information about the following topics in the Theory Guide:

Rotor and Stator Domains

The Mixing Plane Concept

Choosing an Averaging Method

Mixing Plane Algorithm of ANSYS FLUENT

Mass Conservation

Swirl Conservation

Total Enthalpy Conservation

10.3.3. Mesh Setup for a Multiple Moving Reference Frame

Two mesh setup methods are available. Choose the method that is appropriate for your model, noting the restrictions in *Limitations* (p. 522).

- If the boundary between two zones that are in different reference frames is conformal (i.e., the mesh node locations are identical at the boundary where the two zones meet), you can simply create the mesh as usual, with all cell zones contained in the same mesh file. A different cell zone should exist for each portion of the domain that is modeled in a different reference frame. Use an *interior* zone for the boundary between reference frames.
- If the boundary between two zones that are in different reference frames is non-conformal (i.e., the mesh node locations are *not* identical at the boundary where the two zones meet), follow the non-conformal mesh setup procedure described in *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169).

10.3.4. Setting Up a Multiple Moving Reference Frame Problem

10.3.4.1. Setting Up Multiple Reference Frames

To model a problem involving multiple reference frames, perform the following:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The mesh-setup constraints for a moving reference frame listed in *Mesh Setup for a Single Moving Reference Frame* (p. 515) apply to multiple reference frames as well.

1. Select the **Velocity Formulation** to be used in the *General Task Page* (p. 1631): either **Absolute** or **Relative**. (For details, see *Choosing the Relative or Absolute Velocity Formulation* (p. 518).)

   General

   (Note that this step is irrelevant if you are using one of the density-based solution algorithms; these algorithms always use an absolute velocity formulation.)

2. For each cell zone in the domain, specify its translational velocity and/or its angular velocity (ω) and the axis about which it rotates.

   Cell Zones Conditions

   a. If the zone is moving, or if you plan to specify cylindrical velocity or flow-direction components at inlets to the zone, you will need to define the axis of rotation of the frame of reference. In the **Fluid** or **Solid** dialog box, specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** under the **Reference Frame** tab.

   b. In the **Fluid** (*Figure 10.4* (p. 517)) or **Solid** dialog box, enable the **Frame Motion** option. Set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab.

      Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in *Defining Transient Cell Zone and Boundary Conditions* (p. 376), or a UDF macro, described in DEFINE_TRANSIENT_PROFILE. Specifying the individual velocities as either a profile or a UDF allows you to specify a specific input of the frame motion individually. However, you can also specify the frame motion inputs via a single user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

      Note

      If you decide to hook a UDF, then you will no longer have access to the rotation axis origin and direction, or the velocities.

      Details about these inputs are presented in *Inputs for Fluid Zones* (p. 215) and in *Inputs for Solid Zones* (p. 218). Details about the zone motion UDF can be found in DEFINE_ZONE_MOTION in the UDF Manual.

   c. To switch between the MRF and moving mesh models, click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models.

      The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

3. Define the velocity boundary conditions at walls. You can choose to define either an absolute velocity or a velocity relative to the velocity of the adjacent cell zone specified in step 2.

   If the wall is moving at the speed of the moving frame (and hence stationary relative to the moving frame), it is convenient to specify a relative angular velocity of zero. Likewise, a wall that is stationary in the non-moving frame of reference should be given a velocity of zero in the absolute reference frame. Specifying the wall velocities in this manner obviates the need to modify these inputs later if a change is made in the rotational velocity of the fluid zone.

   An example for which you would specify a relative velocity is as follows: If an impeller is defined as **wall-3** and the fluid region within the impeller's radius is defined as **fluid-5**, you would need to specify the angular velocity and axis of rotation for **fluid-5** and then assign **wall-3** a relative velocity of 0. If you later wanted to model a different angular velocity for the impeller, you would need to change only the angular velocity of the fluid region; you would not need to modify the wall velocity conditions.

   Details about these inputs are presented in *Velocity Conditions for Moving Walls* (p. 302).

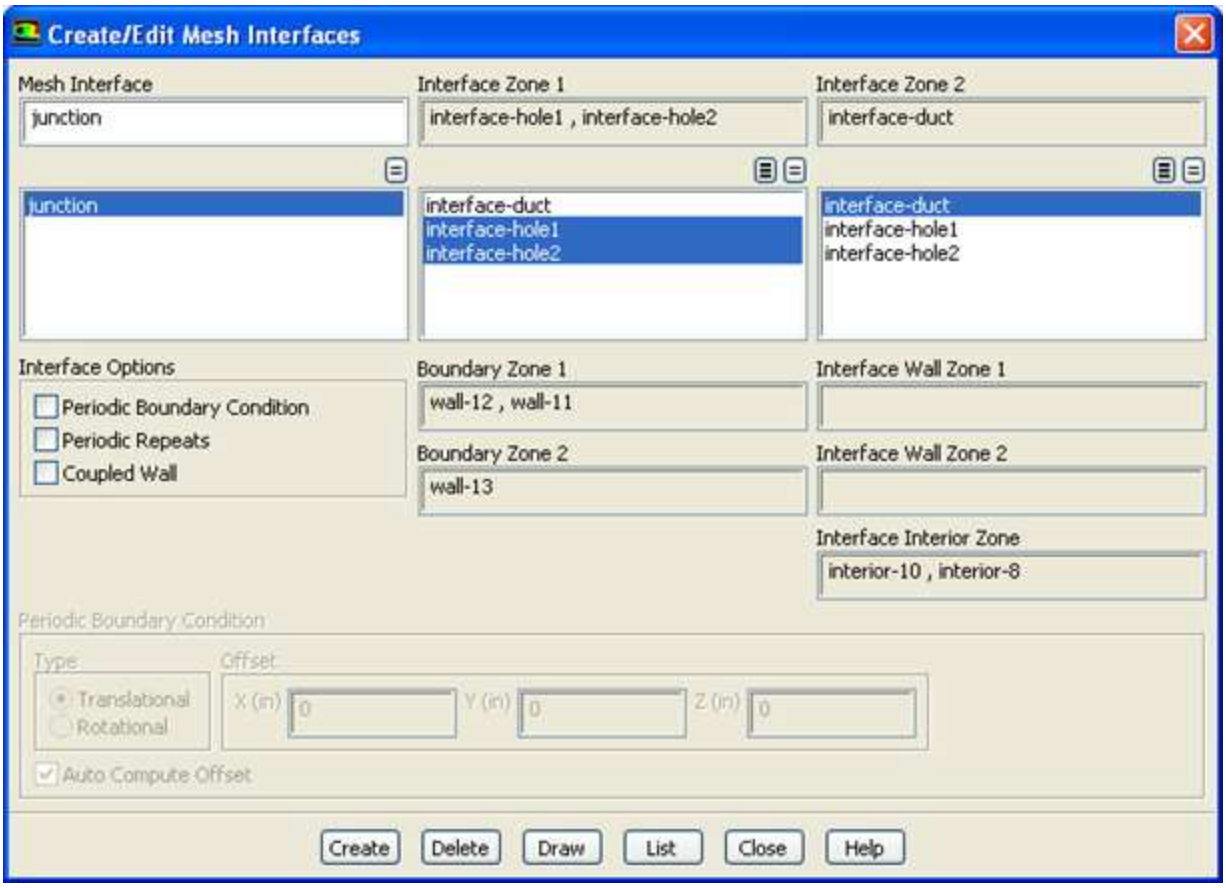
4. Define the boundary conditions at the inlets, as described in *Boundary Conditions* (p. 249). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). Likewise, the total pressure and flow direction can be prescribed in absolute or relative frames for pressure inlets.

   Details about these inputs are presented in *Defining the Flow Direction* (p. 259) and *Defining the Velocity* (p. 266).

5. Define the mesh interfaces in the *Create/Edit Mesh Interfaces Dialog Box* (p. 1876) (*Figure 11.10* (p. 543)).

   Mesh Interfaces → **Create/Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To learn how to use the **Create/Edit Mesh Interfaces** dialog box, see *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169).

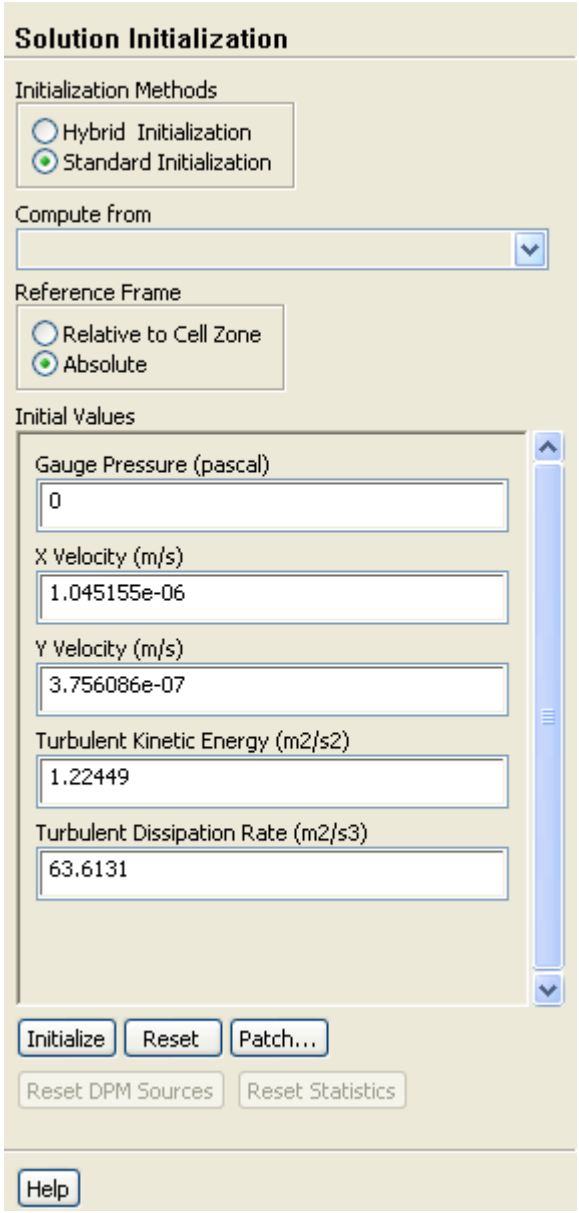
6. Initialize the solution using an absolute frame of reference (*Figure 10.8* (p. 528)).

Solution Initialize

Select the **Absolute** option under **Reference Frame**. If the **Relative to Cell Zone** option is selected, the initial flow field can contain discontinuities, which can cause convergence problems in the first few iterations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.8 The Solution Initialization Task Page for Moving Reference Frames



10.3.4.2. Setting Up the Mixing Plane Model

The model inputs for mixing planes are presented in this section. Only those steps relevant specifically to the setup of a mixing plane problem are listed here. Note that the use of wall and periodic boundaries in a mixing plane model is consistent with their use when the model is not active.

1. Select the **Absolute** or **Relative Velocity Formulation** in the *General Task Page* (p. 1631), when the pressure-based solver is enabled.

   General

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

When the density-based solver is enabled, only the **Absolute Velocity Formulation** can be used with the mixing plane model.

2. For each cell zone in the domain, specify its angular velocity (ω) and the axis about which it rotates.

   Cell Zones Conditions

   a. If the zone is rotating, or if you plan to specify cylindrical-velocity or flow-direction components at inlets to the zone, you will need to define the axis of rotation for the frame of reference. In the **Fluid** dialog box or *Solid Dialog Box* (p. 1802), specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** under the **Reference Frame** tab.

   b. In the **Fluid** (*Figure 10.4* (p. 517)) or **Solid** dialog box, enable the **Frame Motion** option and then set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab.

Important

It is important to define the axis of rotation for the cell zones on *both* sides of the mixing plane interface, including the stationary zone.

3. Define the velocity boundary conditions at walls, as described in step 3 of *Setting Up Multiple Reference Frames* (p. 524).

4. Define the boundary conditions at the inlets, as described in *Boundary Conditions* (p. 249). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). For pressure inlets and mass flow inlets, the specification of the flow direction and total pressure will always be absolute, because the absolute velocity formulation is always used for mixing plane calculations. For a mass flow inlet, you do not need to specify the mass flow rate or mass flux. ANSYS FLUENT will automatically select the **Mass Flux with Average Mass Flux** specification method and set the correct values when you create the mixing plane, as described in *More About Mass Flux and Average Mass Flux* (p. 273).

   Details about these inputs are presented in *Defining the Flow Direction* (p. 259), *Defining the Velocity* (p. 266), and *Inputs at Mass Flow Inlet Boundaries* (p. 271).

Important

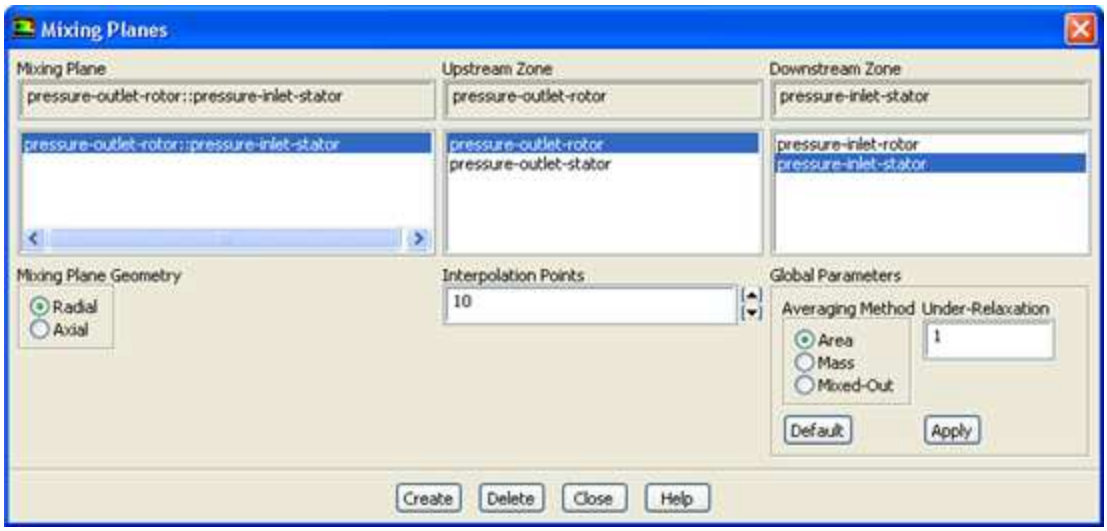
Note that the outlet boundary zone at the mixing plane interface must be defined as a pressure outlet, and the inlet boundary zone at the mixing plane interface must be defined as either a velocity inlet (incompressible flow only), a pressure inlet, or a mass flow inlet. The overall inlet and exit boundary conditions can be any suitable combination permitted by the solver (e.g., velocity inlet, pressure inlet, or mass flow inlet; pressure outlet). Keep in mind, however, that if mass conservation across the mixing plane is important, you need to use a mass flow inlet as the downstream boundary; strict mass conservation is *not* maintained across the mixing plane when you use a velocity inlet or pressure inlet.

5. Define the mixing planes in the *Mixing Planes Dialog Box* (p. 2093) (*Figure 10.9* (p. 530)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define → Mixing Planes...

Figure 10.9 The Mixing Planes Dialog Box



a. Specify the two zones that comprise the mixing plane by selecting an upstream zone in the **Upstream Zone** list and a downstream zone in the **Downstream Zone** list. It is essential that the correct pairs be chosen from these lists (i.e., that the boundary zones selected lie on the mixing plane interface). You can check this by displaying the mesh.

General → Display...

b. (3D only) Indicate the geometry of the mixing plane interface by choosing one of the options under **Mixing Plane Geometry**.

A **Radial** geometry signifies that information at the mixing plane interface is to be circumferentially averaged into profiles that vary in the radial direction, e.g., $p(r), T(r)$. This is the case for axial-flow machines, for example.

An **Axial** geometry signifies that circumferentially averaged profiles are to be constructed that vary in the axial direction, e.g., $p(x), T(x)$. This is the situation for a radial-flow device.

Important

Note that the radial direction is normal to the rotation axis for the fluid zone and the axial direction is parallel to the rotation axis.

c. (3D only) Set the number of **Interpolation Points**. This is the number of radial or axial locations used in constructing the boundary profiles for circumferential averaging. You should choose a number that approximately corresponds to the resolution of the surface mesh in the radial or axial direction. Note that while you can use more points if you wish, the resolution of the boundary profile will only be as fine as the resolution of the surface mesh itself.

In 2D the flow data is averaged over the entire interface to create a profile consisting of a single data point. For this reason you do not need to set the number of **Interpolation Points** or select a **Mixing Plane Geometry** in 2D.

d. Set the **Global Parameters** for the mixing plane.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Select the **Averaging Method**. The **Area** averaging method is the default method. For detailed information about each of the **Area**, **Mass**, or **Mixed-Out** options, see Choosing an Averaging Method in the Theory Guide.

ii. Set the **Under-Relaxation** parameter. It is sometimes desirable to under-relax the changes in boundary values at mixing planes as these may change very rapidly during the early iterations of the solution and cause the calculation to diverge. The changes can be relaxed by specifying an under-relaxation less than 1. The new boundary profile values are then computed using

$$\phi_{new} = \phi_{old} + \alpha \left(\phi_{calculated} - \phi_{old} \right) \tag{10–1}$$

where α is the under-relaxation factor. Once the flow field is established, the value of α can be increased.

iii. Click **Apply** to set the **Global Parameters**. If the **Default** button is visible to the right of the **Apply** button, clicking the **Default** button will return **Global Parameters** back to their default values. The **Default** button will then change to be a **Reset** button. Clicking the **Reset** button will change the **Global Parameters** back to the values that were last applied.

e. Click **Create** to create a new mixing plane. ANSYS FLUENT will name the mixing plane by combining the names of the zones selected as the **Upstream Zone** and **Downstream Zone** and enter the new mixing plane in the **Mixing Plane** list.

If you create an incorrect mixing plane, you can select it in the **Mixing Plane** list and click the **Delete** button to delete it.

10.3.4.2.1. Modeling Options

There are two options available for use with the mixing plane model: a fixed pressure level for incompressible flows, and the swirl conservation described in Swirl Conservation in the Theory Guide.

10.3.4.2.1.1. Fixing the Pressure Level for an Incompressible Flow

For certain turbomachinery configurations, such as a torque converter, there is no fixed-pressure boundary when the mixing plane model is used. The mixing plane model is usually used to model the three interfaces that connect the components of the torque converter. In this configuration, the pressure is no longer fixed. As a result, the pressure may float unbounded, making it difficult to obtain a converged solution.

To resolve this problem, ANSYS FLUENT offers an option for fixing the pressure level. When this option is enabled, ANSYS FLUENT will adjust the gauge pressure field after each iteration by subtracting from it the pressure value in the cell closest to the **Reference Pressure Location** in the *Operating Conditions Dialog Box* (p. 1806).

Important

This option is available only for incompressible flows calculated using the pressure-based solver.

To enable the fixed pressure option, use the `fix-pressure-level` text command:

`define` → `mixing-planes` → `set` → `fix-pressure-level`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.4.2.1.2. Conserving Swirl Across the Mixing Plane

Conservation of swirl is important for applications such as torque converters (Swirl Conservation in the Theory Guide). If you want to enable swirl conservation across the mixing plane, you can use the commands in the `conserve-swirl` text menu:

`define` → `mixing-planes` → `set` → `conserve-swirl`

To turn on swirl conservation, use the `enable?` text command. Once the option is turned on, you can ask the solver to report information about the swirl conservation during the calculation. If you turn on `verbosity?`, ANSYS FLUENT will report for every iteration the zone ID for the zone on which the swirl conservation is active, the upstream and downstream swirl integration per zone area, and the ratio of upstream to downstream swirl integration before and after the correction.

To obtain a report of the swirl integration at every pressure inlet, pressure outlet, velocity inlet, and mass flow inlet in the domain, use the `report-swirl-integration` command. You can use this information to determine the torque acting on each component of the turbomachinery according to Equation 2–22 (in the Theory Guide).

10.3.4.2.1.3. Conserving Total Enthalpy Across the Mixing Plane

One of the options available in the mixing plane model is to conserve total enthalpy across the mixing plane. This is a desirable feature because global parameters such as efficiency are directly related to the change in total enthalpy across a blade row or stage.

The procedure for ensuring conservation of total enthalpy simply involves adjusting the downstream total temperature profile such that the integrated total enthalpy matches the upstream integrated total enthalpy.

If you want to enable total enthalpy conservation, you can use the commands in the `conserve-total-enthalpy` text menu:

`define` → `mixing-planes` → `set` → `conserve-total-enthalpy`

To turn on total enthalpy conservation, use the `enable?` text command. Once the option is turned on, you can ask the solver to report information about the total enthalpy conservation during the calculation. If you turn on `verbosity?`, ANSYS FLUENT will report at every iteration the zone ID for the zone on which the total enthalpy conservation is active, the upstream and downstream heat flux, and the ratio of upstream to downstream heat flux.

10.3.5. Solution Strategies for MRF and Mixing Plane Problems

10.3.5.1. MRF Model

For multiple moving reference frames, follow the guidelines presented in *Solution Strategies for a Single Moving Reference Frame* (p. 519) for a single moving reference frame. Keep in mind that with multiple zones, the possibility exists of interaction between moving and stationary components. This will manifest itself as poor or oscillatory convergence. In such cases, it is strongly recommend that the sliding mesh approach be used to compute the flowfield in order to resolve the unsteady interactions.

10.3.5.2. Mixing Plane Model

It should be emphasized that the mixing plane model is a reasonable approximation so long as there is no significant reverse flow in the vicinity of the mixing plane. If significant reverse flow occurs, the mixing plane will not be a satisfactory model of the actual flow. In a numerical simulation, reverse flow often occurs during

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the early stages of the computation even though the flow at convergence is not reversed. Therefore, it is helpful in these situations to first obtain a provisional solution using *fixed* conditions at the rotor-stator interface. The mixing plane model can then be enabled and the solution run to convergence.

If you are using the mass or mixed-out averaging method and you are experiencing convergence problems in the presence of severe reverse flow, initialize your solution using the default area-averaging method, then switch to mass or mixed-out averaging after the reverse flow dies out.

Under-relaxing the changes in the mixing plane boundary values can also help in troublesome situations. In many cases, setting the under-relaxation factor to a value less than 1 can be helpful. Once the flow field is established, you can gradually increase the under-relaxation factor.

10.3.6. Postprocessing for MRF and Mixing Plane Problems

When you solve a problem using the multiple reference frame or mixing plane model, you can plot or report both absolute and relative velocities. For all velocity parameters (e.g., **Velocity Magnitude** and **Mach Number**), corresponding relative values will be available for postprocessing (e.g., **Relative Velocity Magnitude** and **Relative Mach Number**). These variables are contained in the **Velocity...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Relative values are also available for postprocessing of total pressure, total temperature, and any other parameters that include a dynamic contribution dependent on the reference frame (e.g., **Relative Total Pressure**, **Relative Total Temperature**).

Important

Relative velocities are relative to the translational/rotational velocity of the "reference zone" (specified in the **Reference Values** task page (see *Reference Values Task Page* (p. 1898))). The velocity of the reference zone is the velocity defined in the *Fluid Dialog Box* (p. 1796) for that zone.

When plotting velocity vectors, you can choose to plot vectors in the absolute frame (the default), or you can select **Relative Velocity** in the **Vectors of** drop-down list in the *Vectors Dialog Box* (p. 1970) to plot vectors relative to the translational/rotational velocity of the "reference zone" (specified in the *Reference Values Task Page* (p. 1898)). If you plot relative velocity vectors, you might want to color the vectors by relative velocity magnitude (by choosing **Relative Velocity Magnitude** in the **Color by** list); by default they will be colored by absolute velocity magnitude.

You can also generate a plot of circumferential averages in ANSYS FLUENT. This allows you to find the average value of a quantity at several different radial or axial positions in your model. ANSYS FLUENT computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate. For more information on generating XY plots of circumferential averages, see *XY Plots of Circumferential Averages* (p. 1475).

For details about turbomachinery-specific postprocessing features see *Turbomachinery Postprocessing* (p. 1483).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Modeling Flows Using Sliding and Dynamic Meshes

This chapter describes the setup and use of the sliding and dynamic mesh models in ANSYS FLUENT. To learn more about the theory of sliding meshes in ANSYS FLUENT, see Sliding Mesh Theory in the Theory Guide. For more information about the theory behind dynamic meshes in ANSYS FLUENT, see Dynamic Mesh Theory in the Theory Guide.

Understanding and using the sliding and deforming mesh models is presented in the following sections:

11.1. Introduction
11.2. Sliding Mesh Examples
11.3. The Sliding Mesh Technique
11.4. Sliding Mesh Interface Shapes
11.5. Using Sliding Meshes
11.6. Using Dynamic Meshes

11.1. Introduction

The sliding mesh model allows you to set up a problem in which separate zones move relative to each other. The motion can be translational or rotational. The relative motion of stationary and moving components (e.g., in a rotating machine) will give rise to transient interactions. Often, the transient solution that is sought in a sliding mesh simulation is time-periodic. That is, the transient solution repeats with a period related to the speeds of the moving domains.

The dynamic mesh model allows you to move the boundaries of a cell zone relative to other boundaries of the zone, and to adjust the mesh accordingly. The boundaries can move rigidly with respect to each other (i.e., linear or rotational motion), and/or deform.

When deciding whether to use a sliding mesh verses a dynamic mesh, consider the following:

- Many problems could be solved by either approach.
- If the problem does not involve mesh deformation, the sliding mesh model is recommended, as it is simpler and more efficient.
- The dynamic mesh method must be used if the mesh is deforming, or if the mesh motion is a function of the solution (e.g., the six degree of freedom solver).

For examples of typical sliding mesh and dynamic mesh problems, see Introduction in the Theory Guide.

11.2. Sliding Mesh Examples

When a time-accurate solution (rather than a time-averaged solution) for rotor-stator interaction is desired, you must use the sliding mesh model to compute the unsteady flow field. The sliding mesh model is the most accurate method for simulating flows in multiple moving reference frames, but also the most computationally demanding.

Most often, the unsteady solution that is sought in a sliding mesh simulation is time-periodic. That is, the unsteady solution repeats with a period related to the speeds of the moving domains. However, you can model other types of transients, including translating sliding mesh zones (e.g., two cars or trains passing in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a tunnel, as shown in *Figure 11.1* (p. 536)). Note that the sliding mesh can be modeled using periodic boundaries (*Figure 11.2* (p. 536)) or a circular/cylindrical interface (*Figure 11.3* (p. 536)).

Figure 11.1 Two Passing Trains in a Tunnel

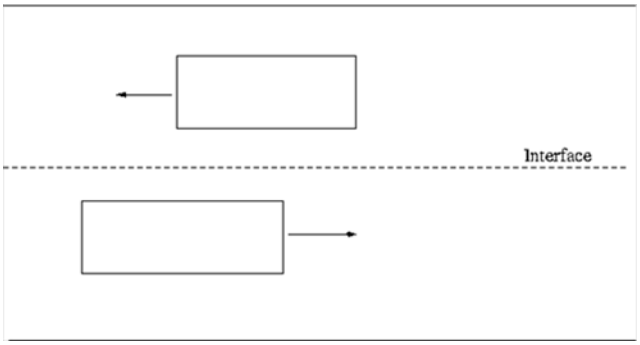


Figure 11.2 Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades)

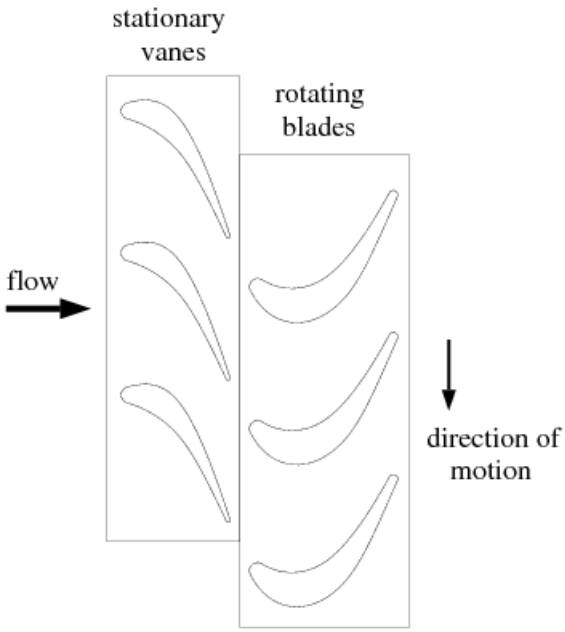
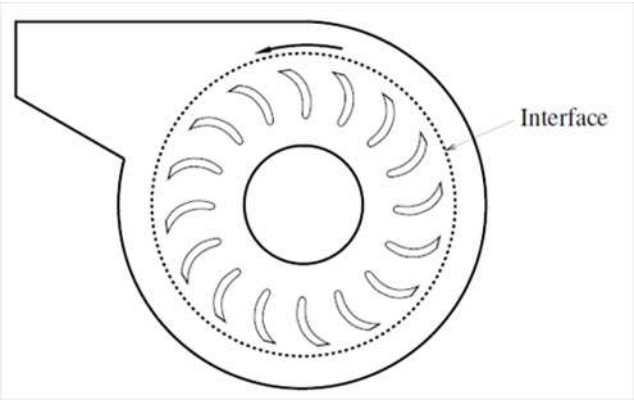


Figure 11.3 Blower



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that for flow situations where there is no interaction between stationary and moving parts (e.g., when there is only a rotor), it is not necessary to use a sliding mesh, and the moving reference frame model is recommended. (See Flow in a Moving Reference Frame for details.) If you are interested in a steady approximation of the interaction, you may use the multiple reference frame model or the mixing plane model, as described in The Multiple Reference Frame Model and The Mixing Plane Model in the Theory Guide.

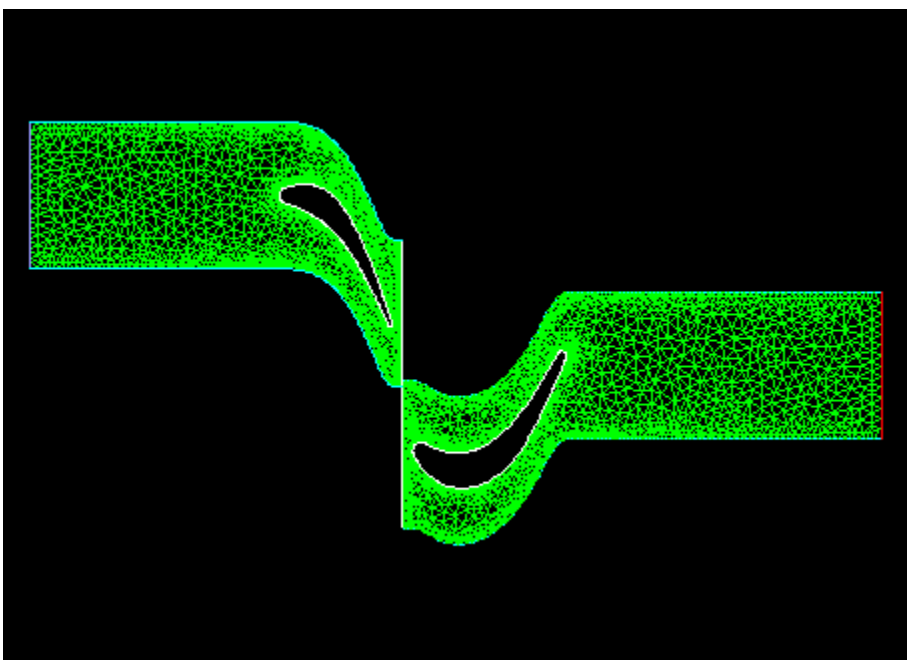
11.3. The Sliding Mesh Technique

In the sliding mesh technique, two or more cell zones are used. (If you generate the mesh in each zone independently, you will need to merge the mesh files prior to starting the calculation, as described in Reading Multiple Mesh/Case/Data Files in the User's Guide.) Each cell zone is bounded by at least one "interface zone" where it meets the opposing cell zone. The interface zones of adjacent cell zones are associated with one another to form a "mesh interface." The two cell zones will move relative to each other along the mesh interface.

During the calculation, the cell zones slide (i.e., rotate or translate) relative to one another along the mesh interface in discrete steps. *Figure 11.4* (p. 537) and *Figure 11.5* (p. 538) show the initial position of two meshes and their positions after some translation has occurred. Note that all non-conformal interfaces will be automatically updated (if necessary) by ANSYS FLUENT when the mesh is updated.

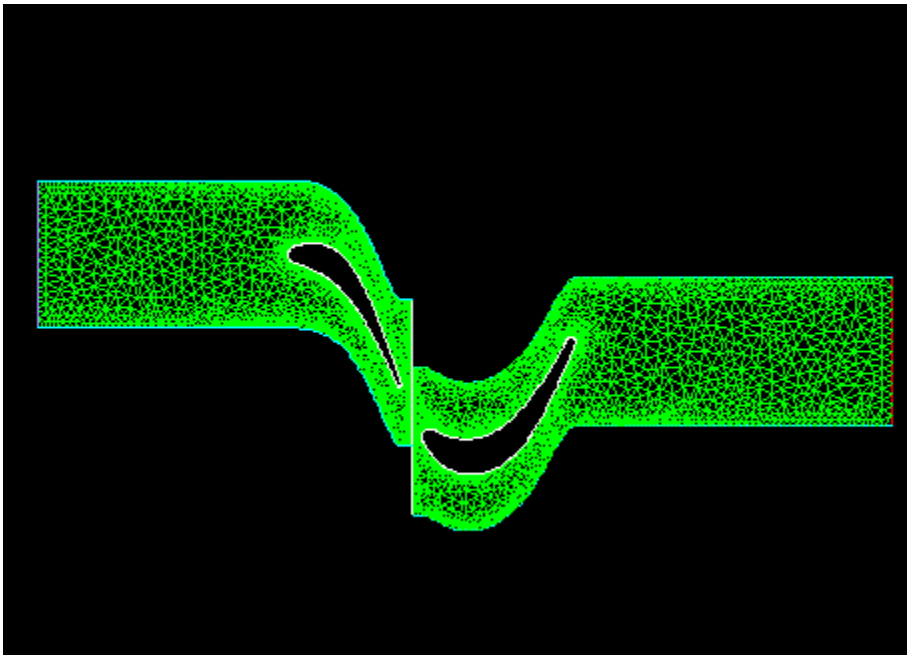
As the rotation or translation takes place, node alignment along the mesh interface is not required. Since the flow is inherently unsteady, a time-dependent solution procedure is required.

Figure 11.4 Initial Position of the Meshes



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

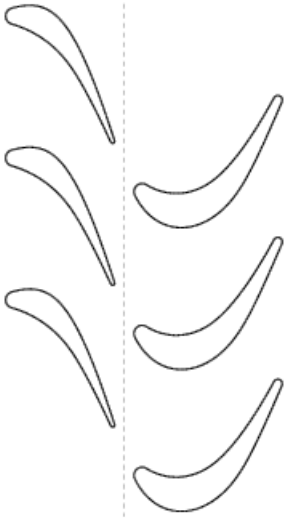
Figure 11.5 Rotor Mesh Slides with Respect to the Stator



11.4. Sliding Mesh Interface Shapes

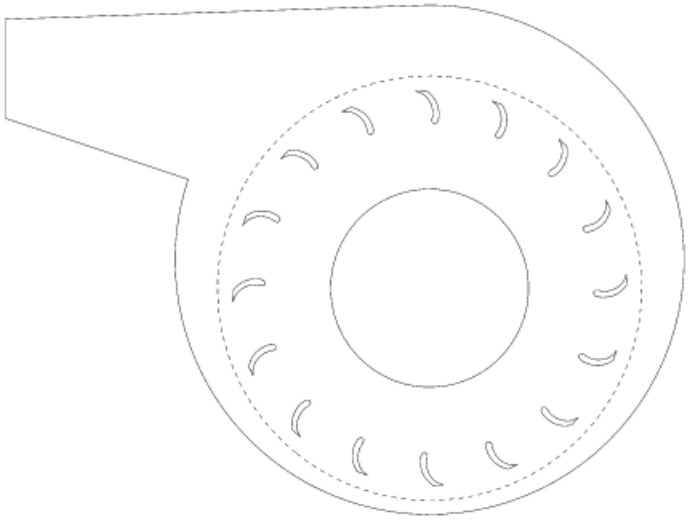
The mesh interface and the associated interface zones can be any shape, provided that the two interface boundaries are based on the same geometry. *Figure 11.6* (p. 538) shows an example with a linear mesh interface and *Figure 11.7* (p. 539) shows a circular-arc mesh interface. (In both figures, the mesh interface is designated by a dashed line.)

Figure 11.6 2D Linear Mesh Interface



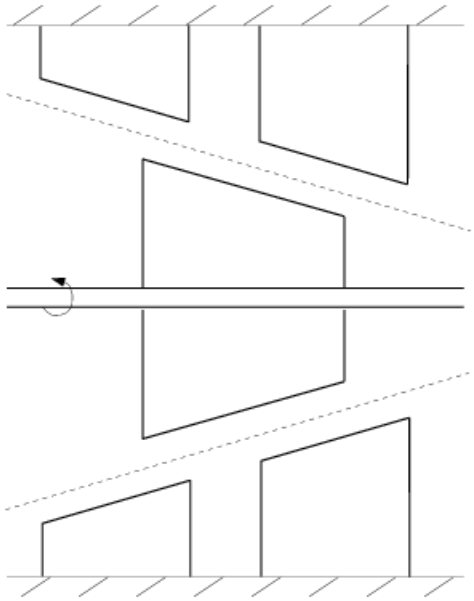
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.7 2D Circular-Arc Mesh Interface



If *Figure 11.6* (p. 538) was extruded to 3D, the resulting sliding interface would be a planar rectangle; if *Figure 11.7* (p. 539) was extruded to 3D, the resulting interface would be a cylinder. *Figure 11.8* (p. 539) shows an example that would use a conical mesh interface. (The slanted, dashed lines represent the intersection of the conical interface with a 2D plane.)

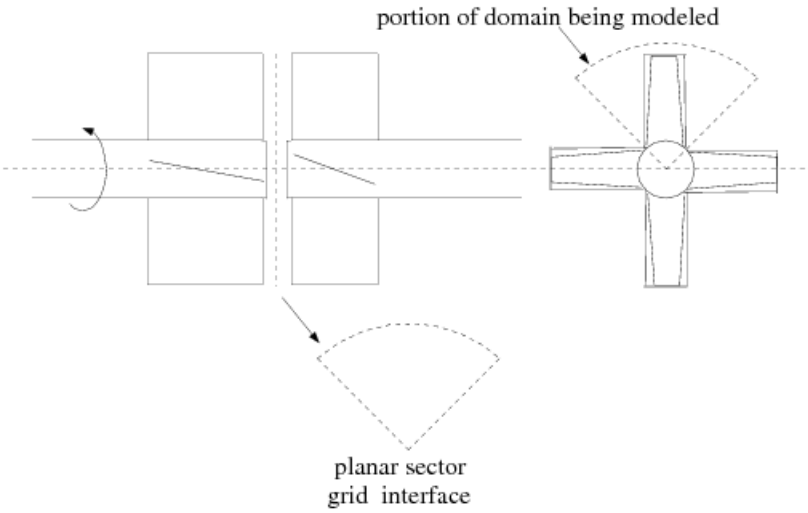
Figure 11.8 3D Conical Mesh Interface



For an axial rotor/stator configuration, in which the rotating and stationary parts are aligned axially instead of being concentric (see *Figure 11.9* (p. 540)), the interface will be a planar sector. This planar sector is a cross-section of the domain perpendicular to the axis of rotation at a position along the axis between the rotor and the stator.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.9 3D Planar-Sector Mesh Interface



11.5. Using Sliding Meshes

This section describes how to use sliding meshes, including restrictions and constraints, problem setup, solution strategies, and postprocessing.

11.5.1. Requirements and Constraints
11.5.2. Setting Up the Sliding Mesh Problem
11.5.3. Solution Strategies for Sliding Meshes
11.5.4. Postprocessing for Sliding Meshes

11.5.1. Requirements and Constraints

Before beginning the problem setup in ANSYS FLUENT, be sure that the mesh you have created meets the following requirements:

- The mesh must have a different cell zone for each portion of the domain that is sliding at a different speed.
- The mesh interface must be situated such that there is no motion normal to it.
- The mesh interface can be any shape (including a non-planar surface, in 3D), provided that the two interface boundaries are based on the same geometry. If there are sharp features in the mesh (e.g., 90-degree angles), it is especially important that both sides of the interface closely follow that feature.
- If you create a single mesh with multiple cell zones, you must be sure that each cell zone has a distinct face zone on the sliding boundary. The face zones for two adjacent cell zones will have the same position and shape, but one will correspond to one cell zone and one to the other. (Note that it is also possible to create a separate mesh file for each of the cell zones, and then merge them as described in *Reading Multiple Mesh/Case/Data Files* (p. 157).)
- If you are modeling a rotor/stator geometry using periodicity, the periodic angle of the mesh around the rotor blade(s) must be the same as that of the mesh around the stationary vane(s).
- All periodic zones must be correctly oriented (either rotational or translational) before you create the mesh interface.
- Note the following limitations if you want to use the periodic repeats option as part of the mesh interface:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - The edges of the second interface zone must be offset from the corresponding edges of the first interface zone by a uniform amount (either a uniform translational displacement or a uniform rotation angle).
  - Some portion of the two interface zones must overlap (i.e., be spatially coincident).
  - The non-overlapping portions of the interface zones must have identical shape and dimensions at all times during the mesh motion.
  - One pair of conformal periodic zones must be adjacent to each of the interface zones. For example, when you calculate just one channel and blade of a fan, turbine, etc., you must have conformal periodics on either side of the interface threads. This will not work with non-conformal periodics.

    Note that for 3D cases, you cannot have more than one pair of conformal periodic zones adjacent to each of the interface zones.
- You must not have a single sliding mesh interface where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (i.e., the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See *Using a Non-Conformal Mesh in ANSYS FLUENT* (p. 169) for information about creating coupled interfaces.

For details about these restrictions and general information about how the sliding mesh model works in ANSYS FLUENT, see *The Sliding Mesh Technique* (p. 537).

11.5.2. Setting Up the Sliding Mesh Problem

The steps for setting up a sliding mesh problem are listed below. (Note that this procedure includes only those steps necessary for the sliding mesh model itself; you will need to set up other models, boundary conditions, etc. as usual.)

1. Enable the appropriate option for modeling transient flow in the *General Task Page* (p. 1631). (See *Performing Time-Dependent Calculations* (p. 1255) for details about the transient modeling capabilities in ANSYS FLUENT.)

   General

2. Set the cell zone conditions for the sliding motion:

   Cell Zones Conditions

   In the *Solid Dialog Box* (p. 1802) or *Fluid Dialog Box* (p. 1796) of each moving fluid or solid zone, enable the **Mesh Motion** option and set the translational and/or rotational velocity under the **Mesh Motion** tab. (Note that a solid zone cannot move at a different speed than an adjacent fluid zone.)

   Important

   Note that simultaneous translation and rotation can be modeled only if the rotation axis and the translation direction are the same (i.e., the origin is fixed).

   Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in *Defining Transient Cell Zone and Boundary Conditions* (p. 376), or a UDF macro, described in DEFINE_TRANSIENT_PROFILE in the UDF Manual. Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the frame motion individually. However, you can also specify the frame motion using a user-defined function. This may

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

Note

If you decide to hook a UDF, then you will no longer have access to the rotation axis origin and direction, or the velocities.

3. Set the boundary conditions for the sliding motion:

   Boundary Conditions

   Change the zone **Type** of the interface zones of adjacent cell zones to **interface** in the *Boundary Conditions Task Page* (p. 1812).

   By default, the velocity of a wall is set to zero relative to the motion of the adjacent mesh. For walls bounding a moving mesh this results in a "no-slip" condition in the reference frame of the mesh. Therefore, you need not modify the wall velocity boundary conditions unless the wall is stationary in the absolute frame, and therefore moving in the relative frame. See *Velocity Conditions for Moving Walls* (p. 302) for details about wall motion.

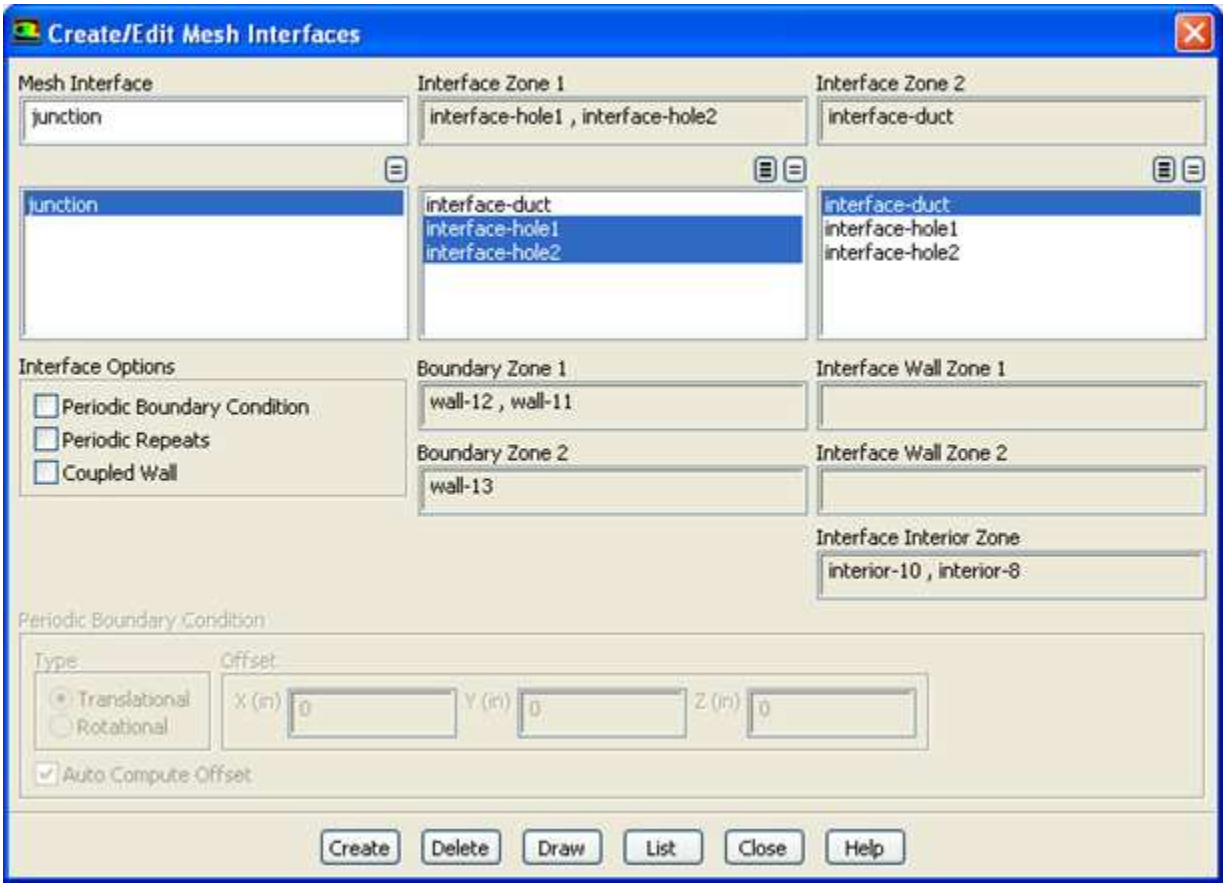
   See *Cell Zone and Boundary Conditions* (p. 205) for details about inputting cell zone and boundary conditions.

4. Define the mesh interfaces in the *Create/Edit Mesh Interfaces Dialog Box* (p. 1876) (*Figure 11.10* (p. 543)).

   Mesh Interfaces → **Create/Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.10 The Create/Edit Mesh Interfaces Dialog Box



a. Enter a name for the interface in the **Mesh Interface** field.

b. Specify the two interface zones that comprise the mesh interface by selecting one or more zones in the **Interface Zone 1** list and one or more zones in the **Interface Zone 2** list. (The order does not matter.)

c. Enable the desired **Interface Options**, if appropriate. There are two options relevant for sliding meshes:

- Enable **Periodic Repeats** when each of the two cell zones has a single pair of conformal periodics adjacent to the interface (see *Figure 6.31* (p. 165)). This option is typically used when simulating the interface between a rotor and stator. See *Non-Conformal Meshes* (p. 161) for further details.

Important

Periodic Repeats is not a valid option when more than one zone is selected in each **Interface Zone**.

- Enable **Coupled Wall** if you would like to model a thermally coupled wall between two fluid zones that share a sliding mesh interface.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the following interfaces are coupled by default:

– the interface between a solid zone and fluid zone

– the interface between a solid zone and solid zone

Therefore, no action is required in the *Create/Edit Mesh Interfaces Dialog Box* (p. 1876) to set up such interfaces.

d. Click **Create** to create a new mesh interface.

For non-periodic interfaces, ANSYS FLUENT will create boundary zones for the interface, which will appear under **Boundary Zone 1** and **Boundary Zone 2**. You can use the *Boundary Conditions Task Page* (p. 1812) to change them to another zone type (e.g., pressure far-field, symmetry, pressure outlet).

If you have enabled the **Coupled Wall** option, ANSYS FLUENT will also create wall interface zones, which will appear under **Interface Wall Zone 1** and **Interface Wall Zone 2**.

If you create an incorrect mesh interface, you can select it in the **Mesh Interface** list and click the **Delete** button to delete it. (Any boundary zones that were created when the interface was created will also be deleted.)

After a **Mesh Interface** is defined, you can select the appropriate mesh interface and click the **Draw** button to display the zones under **Interface Zone 1** and **Interface Zone 2** together as defined by the **Mesh Interface**. This is particularly useful if you want to check the location of the interface zones prior to setting up a mesh interface.

5. Preview the mesh motion using the **Zone Motion** dialog box, which can be opened from the **Run Calculation** task page.

Run Calculation → **Preview Mesh Motion...**

For details about using the **Zone Motion** dialog box, see *Previewing the Dynamic Mesh* (p. 626).

Important

When you have completed the problem setup, you should save an initial case file so that you can easily return to the original mesh position (i.e., the positions before any sliding occurs). The mesh position is stored in the case file, so case files that you save at different times during the transient calculation will contain meshes at different positions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you desire to go from an MRF model setup to a sliding mesh setup, use the following text command:

`mesh` → `modify-zones` → `mrf-to-sliding-mesh`

To successfully switch from an MRF to a sliding mesh, you must provide the ID of the fluid zone. ANSYS FLUENT identifies all the zones belonging to this fluid zone, as well as fluid zones shared in the domain. ANSYS FLUENT then splits these zones into walls, after which the walls are slit converted to interfaces. ANSYS FLUENT then changes the cell zone condition of the fluid zone to **Moving Mesh** in the *Fluid Dialog Box* (p. 1796). The sliding mesh solution tends to be more robust than the MRF solution.

11.5.3. Solution Strategies for Sliding Meshes

You will begin the sliding mesh calculation by initializing the solution (as described in *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240)) and then specifying the time step size and number of time steps in the *Run Calculation Task Page* (p. 1955), as for any other transient calculation. (See *Performing Time-Dependent Calculations* (p. 1255) for details about time-dependent solutions.) Note that the time step size in the initial case file is saved without clicking **Calculate**. ANSYS FLUENT will iterate on the current time step solution until satisfactory residual reduction is achieved, or the maximum number of iterations per time step is reached. When it advances to the next time step, the cell and wall zones will automatically be moved according to the specified translational or rotational velocities (as discussed in the previous section). The new interface-zone intersections will be computed automatically, and resultant interior/periodic/external boundary zones will be updated.

11.5.3.1. Saving Case and Data Files

ANSYS FLUENT's automatic saving of case and data files (see *Automatic Saving of Case and Data Files* (p. 65)) can be used with the sliding mesh model. This provides a convenient way for you to save results at successive time steps for later postprocessing.

Important

You must save a case file each time you save a data file because the mesh position is stored in the case file. Since the mesh position changes with each time step, reading data for a given time step will require the case file at that time step so that the mesh will be in the proper position. You should also save your initial case file so that you can easily return to the mesh's original position to restart the solution if desired.

Important

If you are planning to solve your sliding mesh model in several stages, whereby you run the calculation for some period of time, save case and data files, exit ANSYS FLUENT, start a new ANSYS FLUENT session, read the case and data files, continue the calculation for some time, save case and data files, exit ANSYS FLUENT, and so on, there may be some distortion in the mesh with each subsequent continuation of the calculation. To avoid this problem, you can delete the mesh interface before saving the case file, and then create it again after you read the case file into a new ANSYS FLUENT session.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.5.3.2. Time-Periodic Solutions

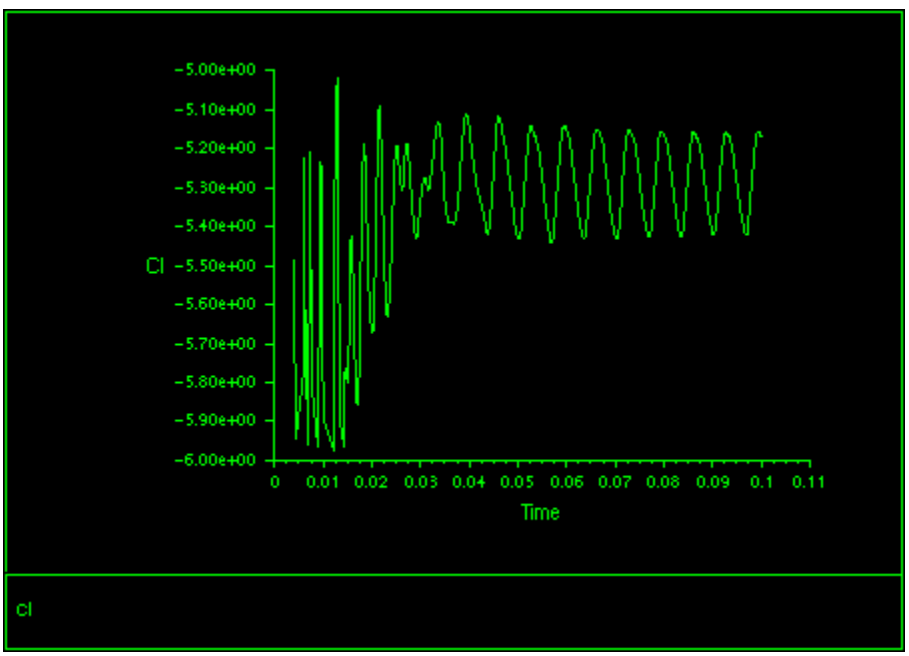
For some problems (e.g., rotor-stator interactions), you may be interested in a time-periodic solution. That is, the startup transient behavior may not be of interest to you. Once this startup phase has passed, the flow will start to exhibit time-periodic behavior. If T is the period of unsteadiness, then for some flow property ϕ at a given point in the flow field:

$$\phi(t) = \phi(t + NT) \; (N = 1, 2, 3, \ldots) \tag{11–1}$$

For rotating problems, the period (in seconds) can be calculated by dividing the sector angle of the domain (in radians) by the rotor speed (in radians/sec): $T = \theta / \Omega$. For 2D rotor-stator problems, $T = P / v_b$, where P is the pitch and v_b is the blade speed. The number of time steps in a period can be determined by dividing the time period by the time step size. When the solution field does not change from one period to the next (for example, if the change is less than 5%), a time-periodic solution has been reached.

To determine how the solution changes from one period to the next, you will need to compare the solution at some point in the flow field over two periods. For example, if the time period is 10 seconds, you can compare the solution at a given point after 22 seconds with the solution after 32 seconds to see if a time-periodic solution has been reached. If not, you can continue the calculation for another period and compare the solutions after 32 and 42 seconds, and so on until you see little or no change from one period to the next. You can also track global quantities, such as lift and drag coefficients and mass flow, in the same manner. *Figure 11.11* (p. 546) shows a lift coefficient plot for a time-periodic solution.

Figure 11.11 Lift Coefficient Plot for a Time-Periodic Solution



The final time-periodic solution is independent of the time steps taken during the initial stages of the solution procedure. You can therefore define "large" time steps in the initial stages of the calculation, since you are not interested in a time-accurate solution for the startup phase of the flow. Starting out with large time steps will allow the solution to become time-periodic more quickly. As the solution becomes time-periodic, however, you should reduce the time step in order to achieve a time-accurate result.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

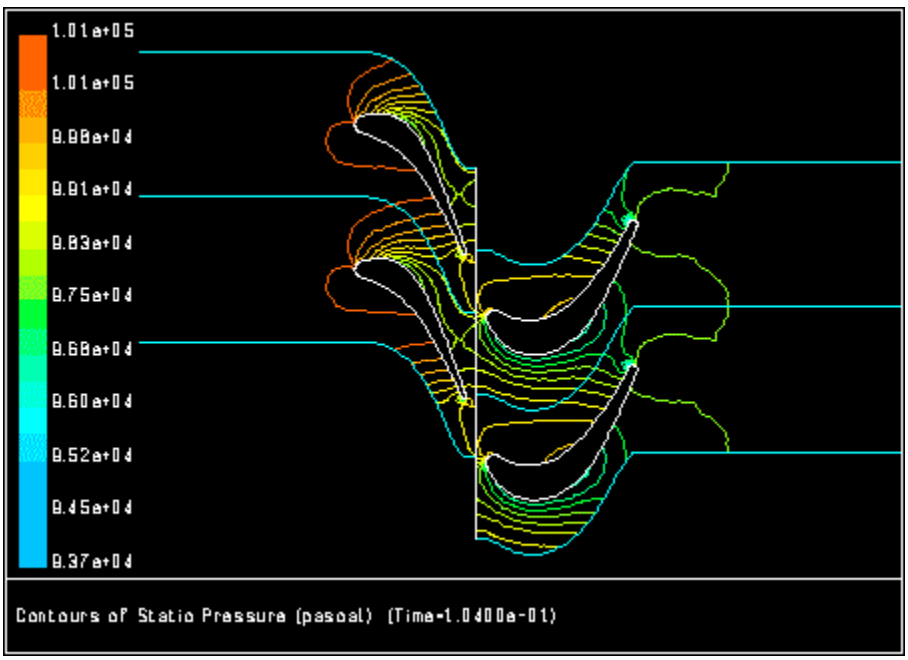
Important

If you are solving with second-order time accuracy, the temporal accuracy of the solution will be affected if you change the time step during the calculation. You may start out with larger time steps, but you should not change the time step by more than 20% during the solution process. You should not change the time step at all during the last several periods to ensure that the solution has approached a time-periodic state.

11.5.4. Postprocessing for Sliding Meshes

Postprocessing for sliding mesh problems is the same as for other transient problems. You will read in the case and data file for the time of interest and display and report results as usual. For spatially-periodic problems, you may want to use periodic repeats (set in the *Views Dialog Box* (p. 2000), as described in *Modifying the View* (p. 1433)) to display the geometry. *Figure 11.12* (p. 547) shows the flow field for the rotor-stator example in *Figure 11.4* (p. 537)) at one instant in time, using 1 periodic repeat.

Figure 11.12 Contours of Static Pressure for the Rotor-Stator Example



When displaying velocity vectors, note that absolute velocities (i.e., velocities in the inertial, or laboratory, reference frame) are displayed by default. You may also choose to display relative velocities by selecting **Relative Velocity** in the **Vectors of** drop-down list in the *Vectors Dialog Box* (p. 1970). In this case, velocities relative to the translational/rotational velocity of the "reference zone" (specified in the *Reference Values Task Page* (p. 1898)) will be displayed. (The velocity of the reference zone is the velocity defined in the *Fluid Dialog Box* (p. 1796) for that zone.)

Note that you cannot create zone surfaces for the intersection boundaries (i.e., the interior/periodic/external zones created from the intersection of the interface zones). You may instead create zone surfaces for the interface zones. Data displayed on these surfaces will be "one-sided". That is, nodes on the interface zones will "see" only the cells on one side of the mesh interface, and slight discontinuities may appear when you plot contour lines across the interface. Note also that, for non-planar interface shapes in 3D, you may see small gaps in your plots of filled contours. These discontinuities and gaps are only graphical in nature. The solution does not have these discontinuities or gaps. To eliminate these discontinuities for postprocessing

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

purposes only, you can use the `define/mesh-interfaces/enforce-continuity-after-bc?` text command, which will ensure that continuity will take precedence over the boundary condition.

You can also generate a plot of circumferential averages in ANSYS FLUENT. This allows you to find the average value of a quantity at several different radial or axial positions in your model. ANSYS FLUENT computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate. For more information on generating XY plots of circumferential averages, see *XY Plots of Circumferential Averages* (p. 1475).

11.6. Using Dynamic Meshes

The steps for setting up a dynamic mesh problem are listed below. (Note that this procedure includes only those steps necessary for the dynamic mesh model itself; you will need to set up other models, cell zone conditions, boundary conditions, etc. as usual.)

1. Enable the appropriate option for modeling transient or steady flow in the *General Task Page* (p. 1631).

   General

   If your problem involves a steady flow, see *Steady-State Dynamic Mesh Applications* (p. 628) for important considerations.

2. Set cell zone conditions and boundary conditions as required in the *Cell Zone Conditions Task Page* (p. 1794) and *Boundary Conditions Task Page* (p. 1812).

   Cell Zone Conditions

   Boundary Conditions

   See *Cell Zone and Boundary Conditions* (p. 205) for information about inputting conditions. The wall velocity is set up automatically when the motion attribute is set for wall zones, so you will not specify wall motion in the **Wall** dialog box. However it is necessary to declare all walls as dynamic zones and specify the motion attributes (see *Specifying the Motion of Dynamic Zones* (p. 613)) in order for wall boundary conditions to be imposed correctly by the solver. Walls which move as part of a dynamic cell zone, but are not declared as dynamic zones, will not impose the correct boundary conditions.

3. Enable the dynamic mesh model, and specify related parameters in the *Dynamic Mesh Task Page* (p. 1877).

   Dynamic Mesh → **Dynamic Mesh**

   See *Setting Dynamic Mesh Modeling Parameters* (p. 550) for details.

4. Create the dynamic zones for your model, using the *Dynamic Mesh Zones Dialog Box* (p. 1889).

   Dynamic Mesh → **Create/Edit...**

   See *Specifying the Motion of Dynamic Zones* (p. 613) for details.

5. You can display the motion of the moving zones with prescribed motion to verify the simulation setup.

   Dynamic Mesh → **Display Zone Motion...**

   See *Previewing the Dynamic Mesh* (p. 626) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If it is a transient simulation, define the events that will occur during the calculation.

   Dynamic Mesh → Events...

   See *Defining Dynamic Mesh Events* (p. 605) for details.

7. Save the case and data.

   File → Write → Case & Data...

8. Preview your dynamic mesh setup (when the motion is a prescribed motion). See *Steady-State Dynamic Mesh Applications* (p. 628) for previewing your steady-state dynamic mesh motion and refer to *Previewing the Dynamic Mesh* (p. 626) for details.

   Dynamic Mesh → Preview Mesh Motion...

9. Specify the pressure-velocity coupling scheme. For transient flow calculations, the PISO algorithm is recommended, as it is the most efficient for such cases (see *PISO* (p. 1216) for details).

10. Use the automatic saving feature to specify the file name and frequency with which case and data files should be saved during the solution process.

    Calculation Activities → Edit... (Autosave Every)

    See *Automatic Saving of Case and Data Files* (p. 65) for details about the use of this feature. This provides a convenient way for you to save results at successive time steps for later postprocessing.

    Important

    You must save a case file each time you save a data file because the mesh position is stored in the case file. Since the mesh position changes with each time step, reading data for a given time step will require the case file at that time step so that the mesh will be in the proper position. You should also save your initial case file so that you can easily return to the mesh's original position to restart the solution if desired.

11. (optional) If you want to create a graphical animation of the mesh over time during the solution procedure, you can use the *Calculation Activities Task Page* (p. 1942) to set up the graphical displays that you want to use in the animation. See *Animating the Solution* (p. 1296) for details.

For additional information, please see the following sections:

11.6.1. Setting Dynamic Mesh Modeling Parameters
11.6.2. Dynamic Mesh Update Methods
11.6.3. In-Cylinder Settings
11.6.4. Six DOF Solver Settings
11.6.5. Implicit Update Settings
11.6.6. Defining Dynamic Mesh Events
11.6.7. Specifying the Motion of Dynamic Zones
11.6.8. Previewing the Dynamic Mesh
11.6.9. Steady-State Dynamic Mesh Applications

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.1. Setting Dynamic Mesh Modeling Parameters

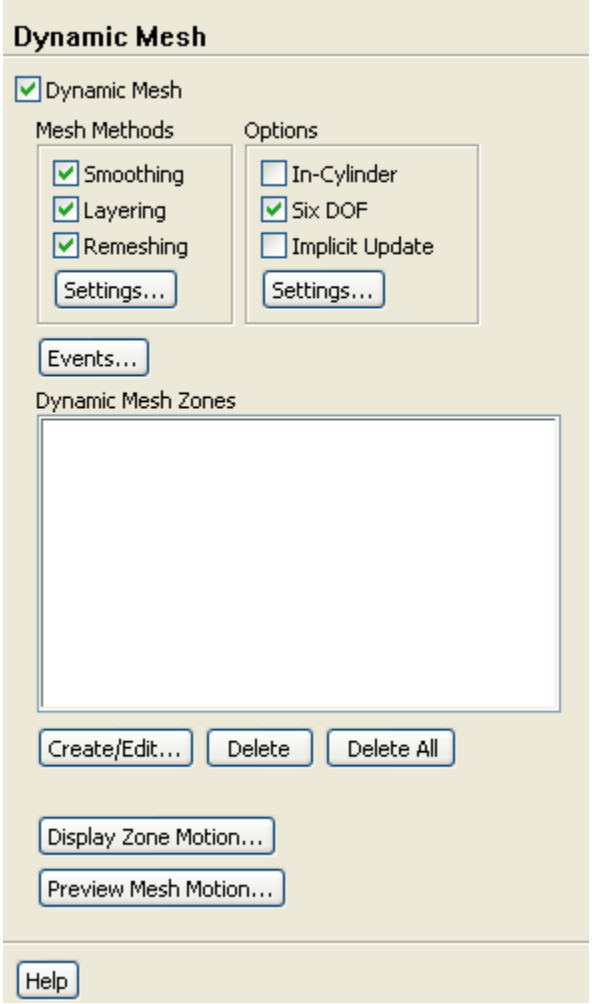
To enable the dynamic mesh model, enable **Dynamic Mesh** in the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.13* (p. 550)).

Dynamic Mesh → **Dynamic Mesh**

Then, enable the appropriate options in the **Options** group box. If you are modeling in-cylinder motion, enable the **In-Cylinder** option. If you are going to use the six degrees of freedom solver, then enable the **Six DOF** option. If you are want to have the dynamic mesh updated during a time step (as opposed to just at the beginning of a time step), then enable the **Implicit Update** option. More information about these options and the related settings can be found in *In-Cylinder Settings* (p. 587), *Six DOF Solver Settings* (p. 601), and *Implicit Update Settings* (p. 603), respectively.

Next, you will need to select the appropriate mesh update methods in the **Mesh Methods** group box, and set the associated parameters, if relevant. See *Dynamic Mesh Update Methods* (p. 551) for details.

Figure 11.13 The Dynamic Mesh Task Page



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.2. Dynamic Mesh Update Methods

Three groups of mesh motion methods are available in ANSYS FLUENT to update the volume mesh in the deforming regions subject to the motion defined at the boundaries:

- smoothing methods
- dynamic layering
- remeshing methods

Note that you can use ANSYS FLUENT's dynamic mesh models in conjunction with hanging node adaption, with the exception of dynamic layering and face remeshing. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

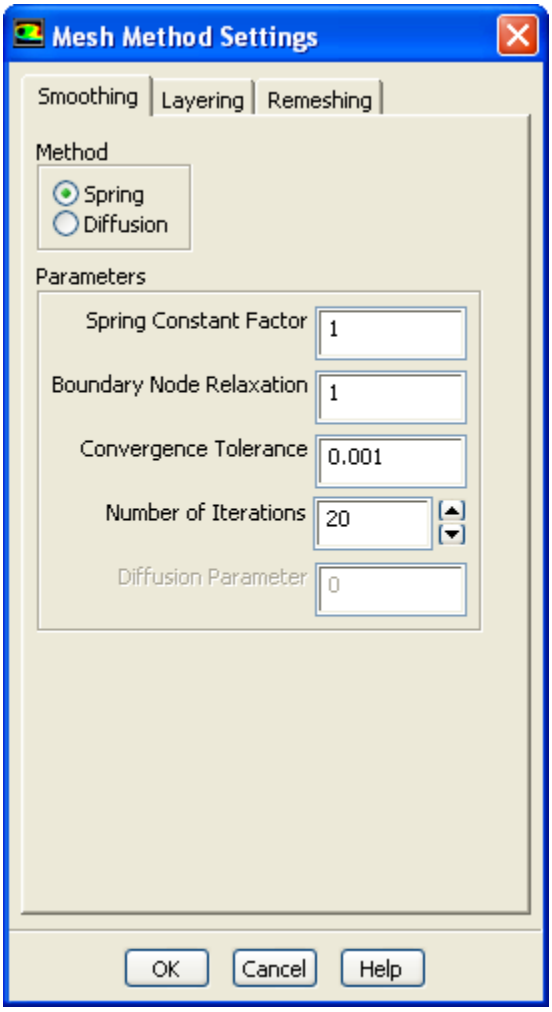
Under **Mesh Methods**, select **Smoothing**, **Layering**, and/or **Remeshing** and click the **Settings...** button. The **Mesh Method Settings** dialog box will open, where you will specify the various settings of the meshing methods. Details about these methods and their applicability to different cases are provided in this section.

11.6.2.1. Smoothing Methods

To enable spring-based or diffusion-based smoothing (or Laplacian smoothing if the 2.5D remeshing method is enabled), enable the **Smoothing** option under **Mesh Methods** in the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.13* (p. 550)). The smoothing method and relevant parameters are specified in the **Smoothing** tab which can be displayed by clicking **Settings...**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.14 The Smoothing Tab in the Mesh Method Settings Dialog Box



11.6.2.1.1. Spring-Based Smoothing

In the spring-based smoothing method, the edges between any two mesh nodes are idealized as a network of interconnected springs. The initial spacings of the edges before any boundary motion constitute the equilibrium state of the mesh. A displacement at a given boundary node will generate a force proportional to the displacement along all the springs connected to the node. Using Hook's Law, the force on a mesh node can be written as

$$\vec{F}_i = \sum_j^{n_i} k_{ij}\left(\Delta \vec{x}_j - \Delta \vec{x}_i\right) \qquad \textbf{(11–2)}$$

where $\Delta \vec{x}_i$ and $\Delta \vec{x}_j$ are the displacements of node i and its neighbor j, n_i is the number of neighboring nodes connected to node i, and k_{ij} is the spring constant (or stiffness) between node i and its neighbor j. The spring constant for the edge connecting nodes i and j is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k_{ij} = \frac{1}{\sqrt{\left|\vec{x}_i - \vec{x}_j\right|}} \tag{11–3}$$

At equilibrium, the net force on a node due to all the springs connected to the node must be zero. This condition results in an iterative equation such that

$$\Delta \vec{x}_i^{m+1} = \frac{\sum_j^{n_i} k_{ij} \Delta \vec{x}_j^m}{\sum_j^{n_i} k_{ij}} \tag{11–4}$$

Since displacements are known at the boundaries (after boundary node positions have been updated), *Equation 11–4* (p. 553) is solved using a Jacobi sweep on all interior nodes. At convergence, the positions are updated such that

$$\vec{x}_i^{n+1} = \vec{x}_i^n + \Delta \vec{x}_i^{m,converged} \tag{11–5}$$

where $n+1$ and n are used to denote the positions at the next time step and the current time step, respectively. The spring-based smoothing is shown in *Figure 11.15* (p. 553) and *Figure 11.16* (p. 554) for a cylindrical cell zone where one end of the cylinder is moving.

Figure 11.15 Spring-Based Smoothing on Interior Nodes: Start

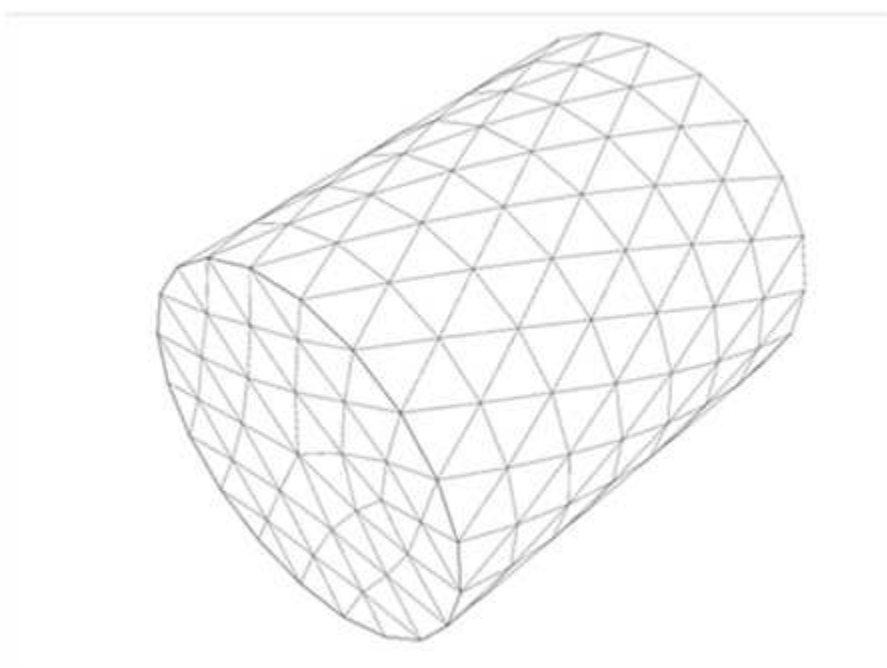
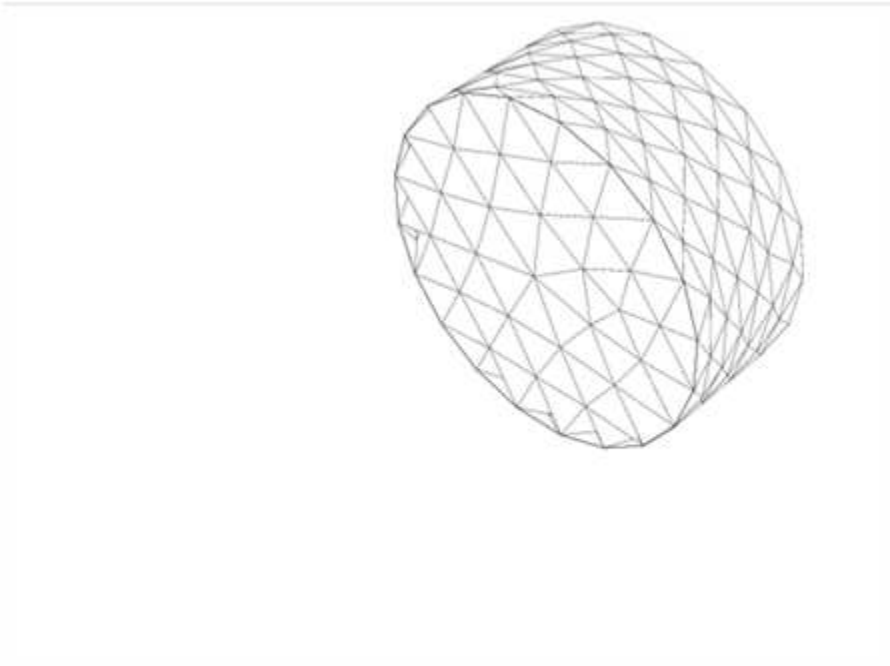


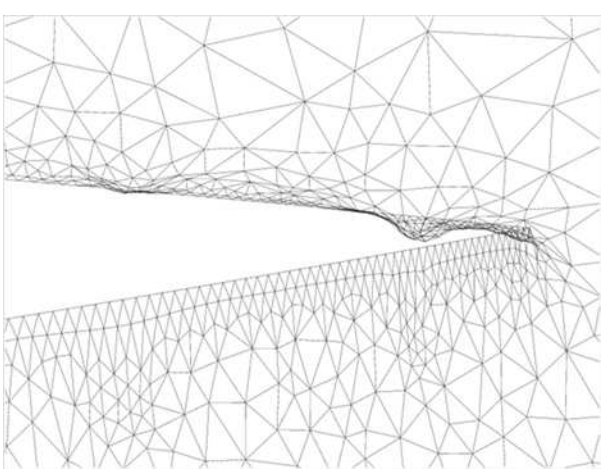
Figure 11.16 Spring-Based Smoothing on Interior Nodes: End



You can control the spring stiffness by adjusting the value of the **Spring Constant Factor** between 0 and 1. A value of 0 indicates that there is no damping on the springs, and boundary node displacements have more influence on the motion of the interior nodes. A value of 1 imposes the default level of damping on the interior node displacements as determined by solving *Equation 11–4* (p. 553).

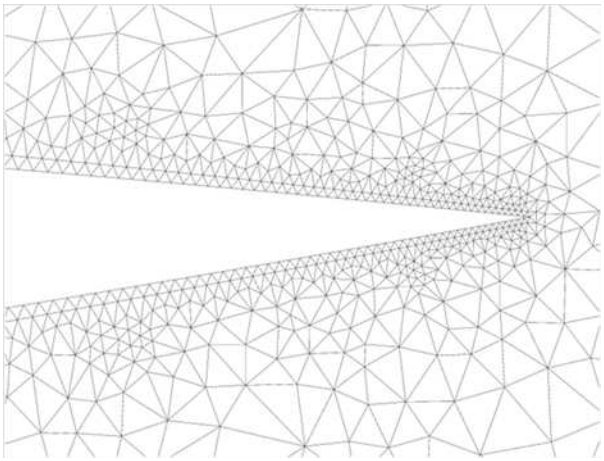
The effect of the **Spring Constant Factor** is illustrated in *Figure 11.17* (p. 554) and *Figure 11.18* (p. 555), which show the trailing edge of a NACA-0012 airfoil after a counter-clockwise rotation of 2.3 $^{\circ}$ and the mesh is smoothed using the spring-based smoother but limited to 20 iterations. Degenerate cells (*Figure 11.17* (p. 554)) are created with the default value of 1 for the **Spring Constant Factor**. However, the original mesh distribution (*Figure 11.18* (p. 555)) is recovered if the **Spring Constant Factor** is set to 0 (i.e., no damping on the displacement of nodes on the airfoil surface).

Figure 11.17 Effect of a Spring Constant Factor of 1 on Interior Node Motion



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.18 Effect of a Spring Constant Factor of 0 on Interior Node Motion



If your model contains deforming boundary zones, you can use the **Boundary Node Relaxation** to control how the node positions on the deforming boundaries are updated. On deforming boundaries, the node positions are updated such that

$$\vec{x}^{\,n+1} = \vec{x}^{\,n} + \beta \Delta \vec{x}^{\,m,converged}_{spring} \tag{11–6}$$

where β is the **Boundary Node Relaxation**. A value of 0 prevents deforming boundary nodes from moving (equivalent to turning off smoothing on deforming boundary zones) and a value of 1 indicates no under-relaxation.

You can control the solution of *Equation 11–4* (p. 553) using the values of **Convergence Tolerance** and **Number of Iterations**. ANSYS FLUENT solves *Equation 11–4* (p. 553) iteratively during each time step until one of the following criteria is met:

- The specified number of iterations has been performed.
- The solution is converged for that time step:

$$\left(\frac{\Delta \vec{x}^{\,m}_{rms}}{\Delta \vec{x}^{\,1}_{rms}} \right) < \text{convergence tolerance} \tag{11–7}$$

  where $\Delta \vec{x}^{\,1}_{rms}$ is the interior and deforming nodes RMS displacement at the first iteration.

11.6.2.1.1.1. Applicability of the Spring-Based Smoothing Methods

You can use the spring-based smoothing method to update any cell or face zone whose boundary is moving or deforming.

For non-tetrahedral cell zones (non-triangular in 2D), the spring-based method is recommended when the following conditions are met:

- The boundary of the cell zone moves predominantly in one direction (i.e., no excessive anisotropic stretching or compression of the cell zone).
- The motion is predominantly normal to the boundary zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If these conditions are not met, the resulting cells may have high skewness values, since not all possible combinations of node pairs in non-tetrahedral cells (or non-triangular in 2D) are idealized as springs.

By default, spring-based smoothing on non-triangular or non-tetrahedral cell zones are turned off. If you want to use spring-based smoothing on all cell shapes, you can turn on the model for these zones using the `spring-on-all-shapes?` text interface command:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `spring-on-all-shapes?`

11.6.2.1.2. Diffusion-Based Smoothing

In the diffusion-based smoothing method, the mesh motion is governed by the diffusion equation:

$$\nabla \cdot (\gamma \nabla \vec{u}) = 0 \tag{11–8}$$

where $\vec{u}$ is the mesh displacement velocity. The boundary conditions to *Equation 11–8* (p. 556) are obtained from the user prescribed or computed (Six DOF) boundary motion. On deforming boundaries, the boundary conditions are such that the mesh motion is tangent to the boundary (that is, the normal velocity component vanishes). The Laplace equation *Equation 11–8* (p. 556) then describes how the prescribed boundary motion diffuses into the interior of the deforming mesh.

The diffusion coefficient γ in *Equation 11–8* (p. 556) can be used to control how the boundary motion affects the interior mesh motion. A constant coefficient means that the boundary motion diffuses uniformly throughout the mesh. With a nonuniform diffusion coefficient, mesh nodes in regions with high diffusivity tend to move together (that is, with less relative motion). The diffusion coefficient implemented in ANSYS FLUENT is of the form

$$\gamma = \frac{1}{d^{\alpha}} \tag{11–9}$$

where d is a normalized boundary distance and $\alpha \geq 0$ is a user input parameter. Increasing the diffusivity near boundaries will allow you to absorb more mesh motion away from the boundaries and better preserve the mesh quality near the boundaries. This is particularly helpful when moving boundaries have pronounced geometrical features, such as sharp corners, and when the boundary motion is predominantly rotational.

The boundary distance d in *Equation 11–9* (p. 556) is either the normalized standard wall distance (nearest distance to all walls, as calculated for the turbulence models) or a generalized boundary distance, where all boundaries (including inlets, outlets, symmetry, and periodic boundaries) not declared as deforming are included. In either case, the distance is normalized by the largest distance found in all deforming cell zones.

The vector *Equation 11–8* (p. 556) is discretized using ANSYS FLUENT's standard finite volume method, and the resulting matrix is solved iteratively using the algebraic multigrid (AMG) solver. If the generalized boundary distance is used as the distance d in *Equation 11–9* (p. 556), then an additional scalar equation has to be solved to evaluate the boundary distance. The cell centered solution for the displacement velocity u is interpolated onto the nodes using inverse distance weighted averaging, and the node positions are updated according to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{x}_{new} = \vec{x}_{old} + \vec{u}\,\Delta t \qquad \textbf{(11–10)}$$

Computationally, diffusion-based smoothing is generally more costly than spring-based smoothing. But it tends to produce better quality meshes than spring-based smoothing and often allows larger boundary deformations before breaking down. *Figure 11.19* (p. 557) and *Figure 11.20* (p. 557) show a mesh before and after rotating the boundary by 45 degrees, using diffusion-based smoothing. With spring-based smoothing, the same mesh shows degenerated cells after a rotation of 40 degrees (*Figure 11.21* (p. 558)).

Figure 11.19 The Initial Mesh

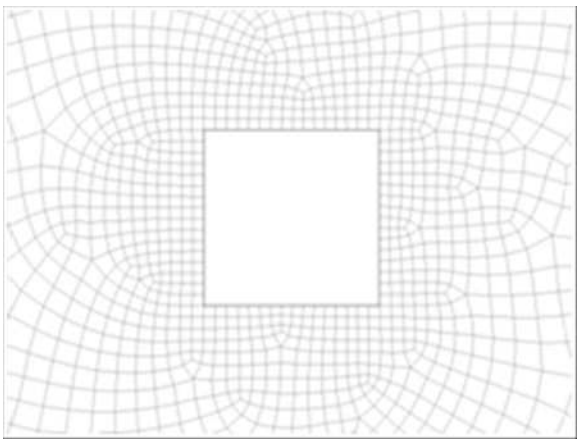
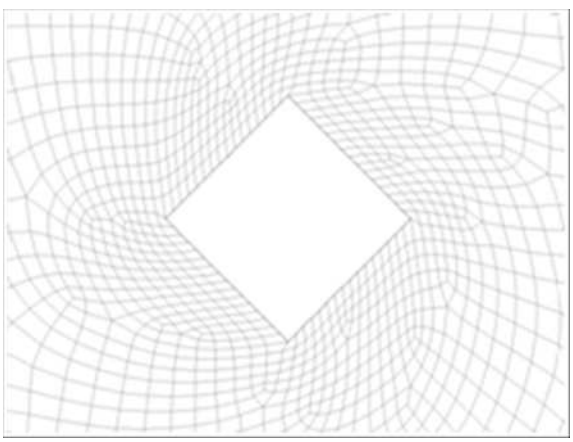
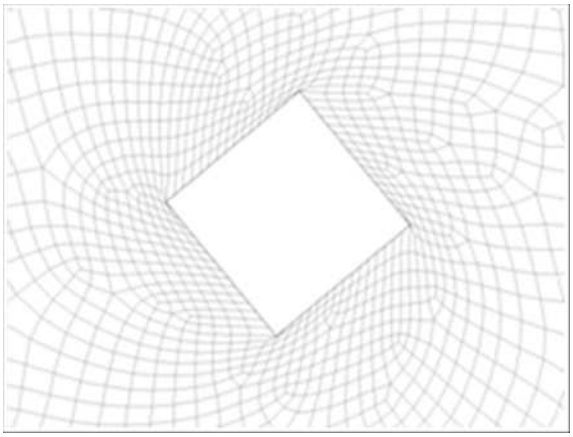


Figure 11.20 Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.21 Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing



It should be noted that with diffusion-based smoothing the interior mesh motion is governed by the solution to *Equation 11–8* (p. 556) and the prescribed boundary motion, and not by mesh irregularities. Poor quality elements or mesh defects are not smoothed out by this method, but rather move together with the pre-computed (at the begin of each mesh update) displacement velocity $\vec{u}$.

It is also worth noting that the nature of the diffusion equation is such that the resulting solution, the displacement velocity $\vec{u}$, depends on the dimensionality of the problem and the type of boundary motion prescribed. For more translational (piston-type) boundary motions, α in *Equation 11–9* (p. 556) is an effective parameter to control where the boundary motion should get absorbed in the interior. For rotational boundary motions the decay of the solution away from the boundary is such that it is often difficult to move the mesh even close to the boundary sufficiently with the boundary. For this reason, mesh smoothing can handle translational boundary motions generally much better than rotational motions.

When you want to specify spatially varying diffusion, you control the diffusion coefficient γ (in *Equation 11–9* (p. 556)) primarily by adjusting the **Diffusion Parameter** α. A range of 0 to 2 has been shown to be of practical use. A value of 0 (default) implies $\gamma = 1$ and a uniform diffusion of the boundary motion into the interior of the mesh. Higher values of alpha will cause the mesh closer to the boundary to move with the boundary, and the boundary deformation is felt farther away from the boundary. *Figure 11.22* (p. 559) and *Figure 11.23* (p. 559) illustrate the effect of the **Diffusion Parameter** α on the resulting mesh for a translational (piston-type) boundary motion. In this example, an initially uniformly meshed square domain is deformed by moving the left boundary to the right.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.22 Effect of a Diffusion Parameter of 0 on Interior Node Motion

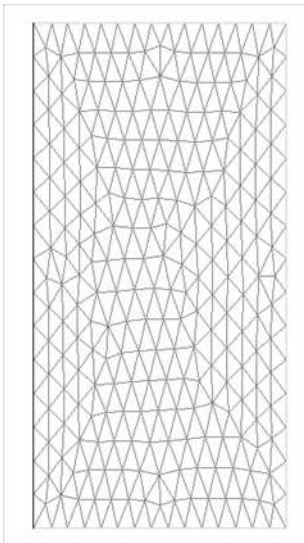
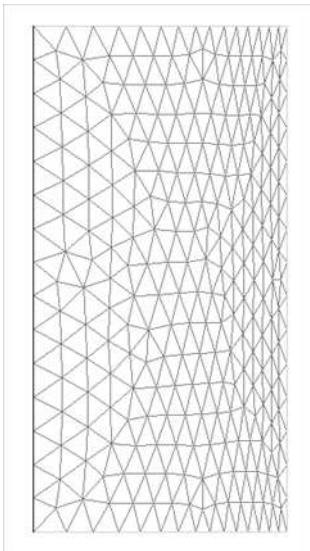


Figure 11.23 Effect of a Diffusion Parameter of 1 on Interior Node Motion



For rotational boundary motions, a value of 1.5 for the **Diffusion Parameter** α is recommended as a good starting point.

Two different boundary distance evaluations are available for the calculation of γ. By default, ANSYS FLUENT uses the standard wall distance for d. All other boundaries (such as inlets, outlets, symmetry, and periodic boundaries) do not contribute to the boundary distance. An alternative generalized boundary distance is available, where all boundaries not declared as deforming are taken into consideration. In the example shown in *Figure 11.22* (p. 559) and *Figure 11.23* (p. 559), only the left boundary is used for the distance calculation. The generalized boundary distance is selected using the text interface command:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `boundary-distance-method`

Although it should in most cases not be necessary, the accuracy of the solution to the diffusion equation governing the mesh motion can be controlled by selecting the maximum number of iterations, `max-iter`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(by default 20), and the relative residual tolerance, `relative-convergence-tolerance` (by default 1.0e-4), in the text interface:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `max-iter`

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `relative-convergence-tolerance`

11.6.2.1.2.1. Applicability of Diffusion-Based Smoothing Method

Diffusion-based mesh smoothing is an alternative method to spring-based smoothing. It is by default available for all element types, and you can use it to update any cell zone whose boundaries are moving or deforming.

Diffusion-based smoothing is computationally more expensive than spring-based smoothing, but likely results in better mesh quality (especially for non-tetrahedral cell zones (non-triangular in 2D)) and generally allows for larger boundary deformations before breaking down.

Similar to spring-based smoothing, diffusion-based mesh smoothing can handle translational boundary deformations much better than rotational motions.

11.6.2.1.3. Laplacian Smoothing Method

Laplacian smoothing is the most commonly used and the simplest mesh smoothing method. This method adjusts the location of each mesh vertex to the geometric center of its neighboring vertices. This method is computationally inexpensive but it does not guarantee an improvement on mesh quality, since repositioning a vertex by Laplacian smoothing can result in poor quality elements. To overcome this problem, ANSYS FLUENT only relocates the vertex to the geometric center of its neighboring vertices if and only if there is an improvement in the mesh quality (i.e., the skewness has been improved).

This improved Laplacian smoothing can be enabled on deforming boundaries only (i.e., the zone with triangular elements in 3D and zones with linear elements in 2D). The computation of the node positions works as follows:

$$\overline{\vec{x}_i^{\,m}} = \frac{\sum_j^{n_i} \vec{x}_j^{\,m}}{n_i} \qquad \textbf{(11–11)}$$

where $\overline{\vec{x}_i^{\,m}}$ is the averaged node position of node i at iteration m, $\vec{x}_j^{\,m}$ is the node position of neighbor node of $\vec{x}_i^{\,m}$ at iteration m, and n_i is the number nodes neighboring node i. The new node position $\vec{x}_i^{\,m+1}$ is then computed as follows:

$$\vec{x}_i^{\,m+1} = \vec{x}_i^{\,m}(1-\beta) + \overline{\vec{x}_i^{\,m}}\beta \qquad \textbf{(11–12)}$$

where β is the boundary node relaxation factor.

This update only happens if the maximum skewness of all faces adjacent to $\vec{x}_i^{\,m+1}$ is improved in comparison to $\vec{x}_i^{\,m}$.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that for 2.5D remeshing method modeling (3D flows only), you can only change the **Boundary Node Relaxation** and the **Number of Iterations**. Note that the **Number of Iterations** is used for both spring-based and Laplacian smoothing. The **Boundary Node Relaxation** is used differently by ANSYS FLUENT when the 2.5D remeshing method model is used. On deforming boundaries, the node positions are updated such that

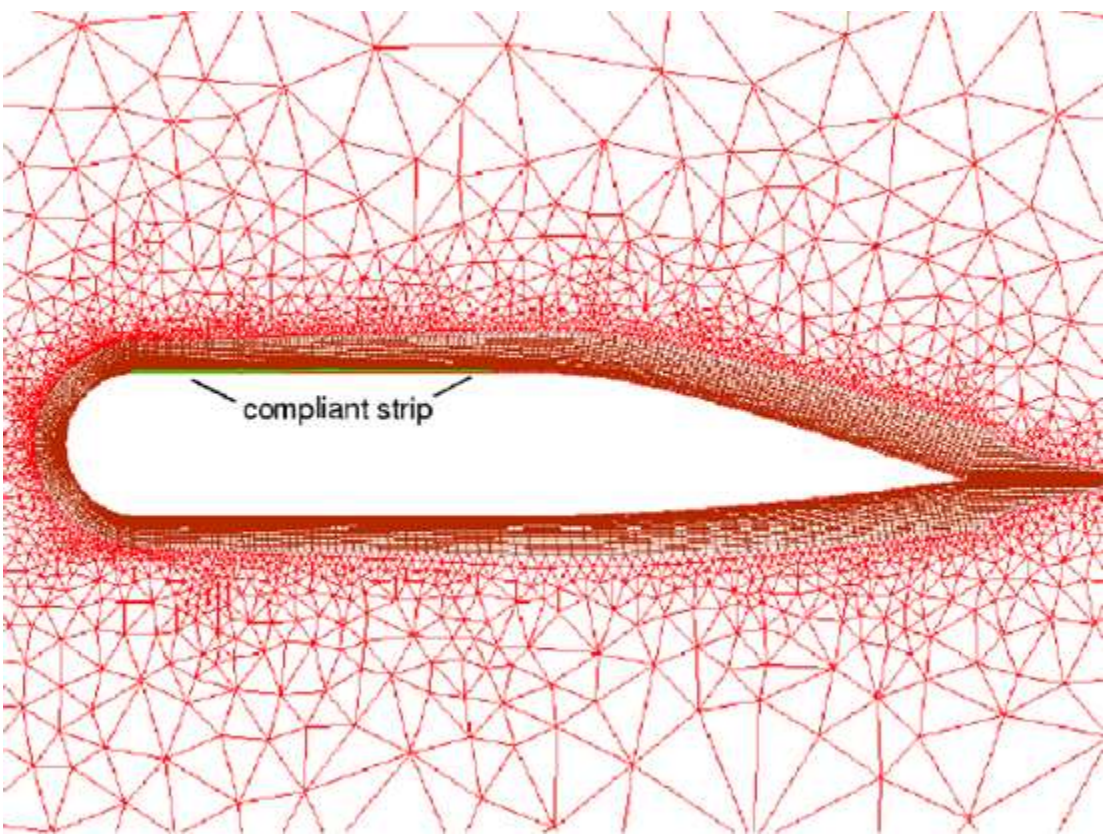
$$\vec{x}^{\,n+1} = \vec{x}^{\,n} + \beta \Delta \vec{x}^{\,m}_{Laplacian} \qquad \textbf{(11–13)}$$

11.6.2.1.4. Boundary Layer Smoothing Method

The boundary layer smoothing method is used to deform the boundary layer during a moving-deforming mesh simulation. For cases that have a **Mesh Motion UDF** (see User-Defined Motion in the User's Guide) applied to a face zone with adjacent boundary layers, the boundary layer will deform according to the UDF that is applied to the face zone. This smoothing method preserves the height of each boundary layer and can be applied to boundary layer zones of all mesh types (wedges and hexahedra in 3D, quadrilaterals in 2D).

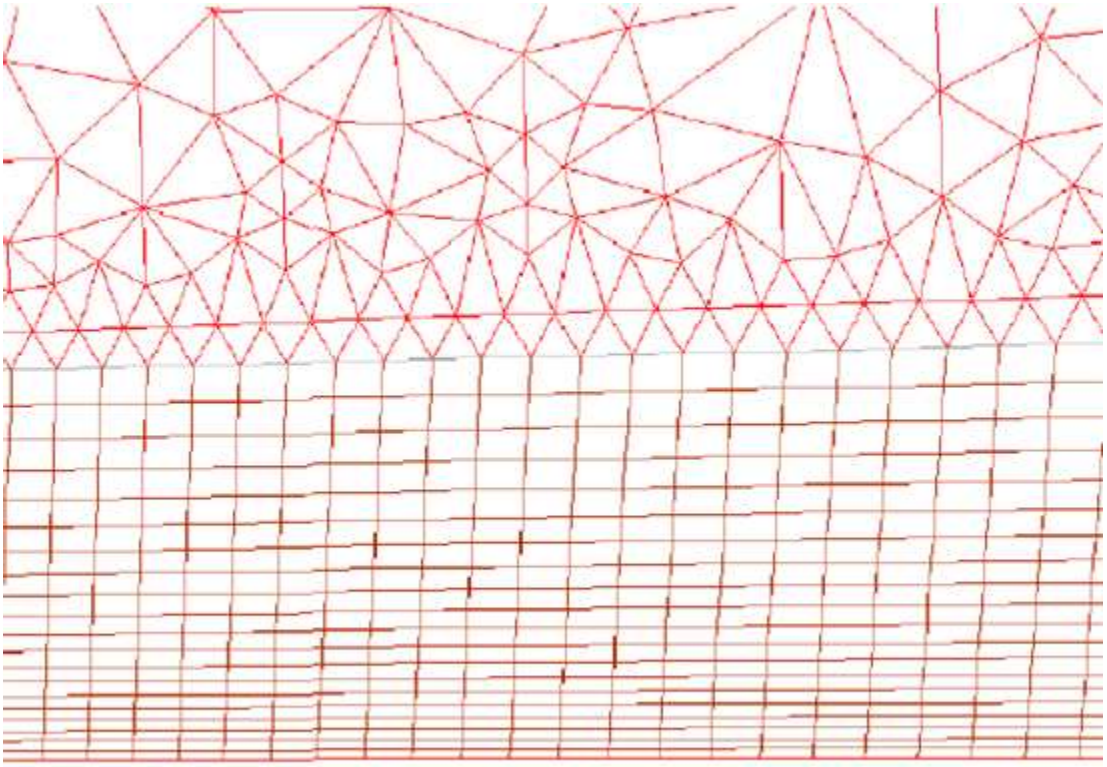
Consider the example below, where a UDF of the form `DEFINE_GRID_MOTION` provides the moving-deforming mesh model with the locations of the nodes located on the compliant strip on an idealized airfoil. The node motion varies sinusoidally (*Figure 11.26* (p. 563) and *Figure 11.27* (p. 564)), both in time and space as seen by the deformation of the face zone and the respective boundary layer. A deforming flag is set on the adjacent cell zone, such that the cells adjacent to the deforming wall will also be deformed, in order to avoid skewness. Compare the original mesh (*Figure 11.24* (p. 561)) prior to applying the **Mesh Motion UDF** to that mesh whose boundary layer has been deformed (*Figure 11.26* (p. 563)).

Figure 11.24 The Mesh Prior to Applying Boundary Layer Smoothing



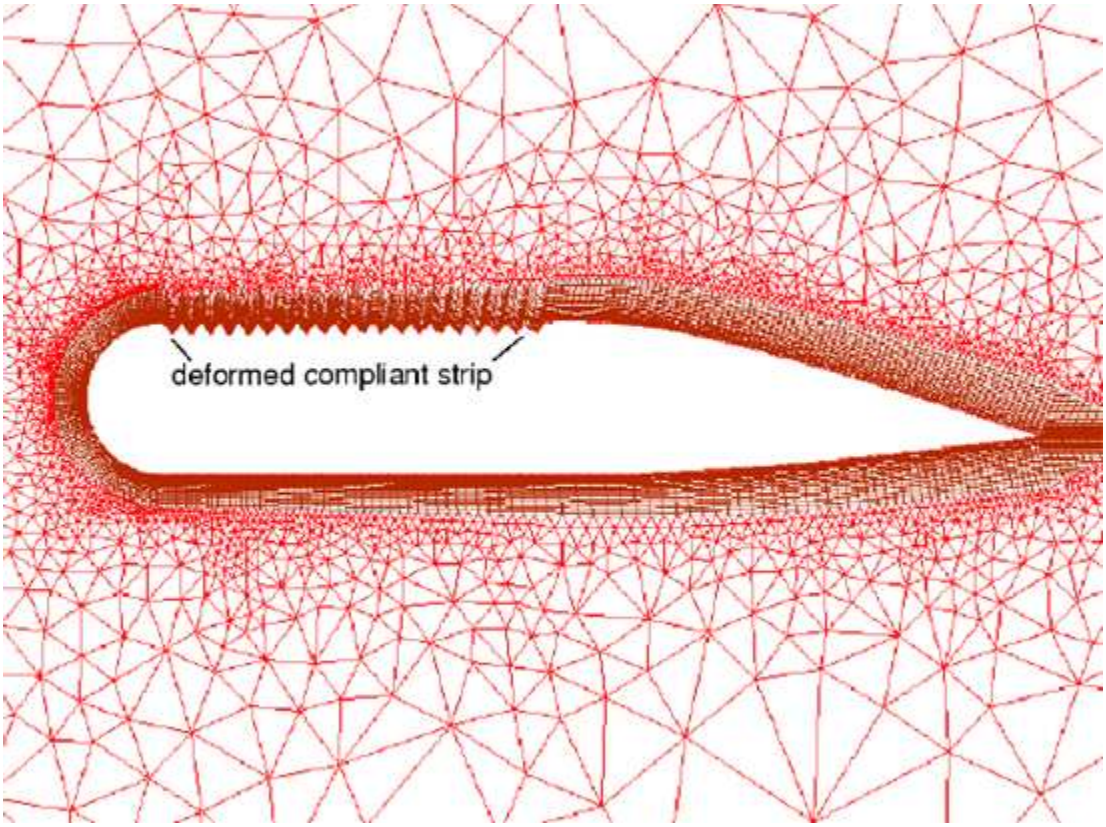
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.25 Zooming into the Mesh of the Compliant Strip Prior to Applying Boundary Layer Smoothing



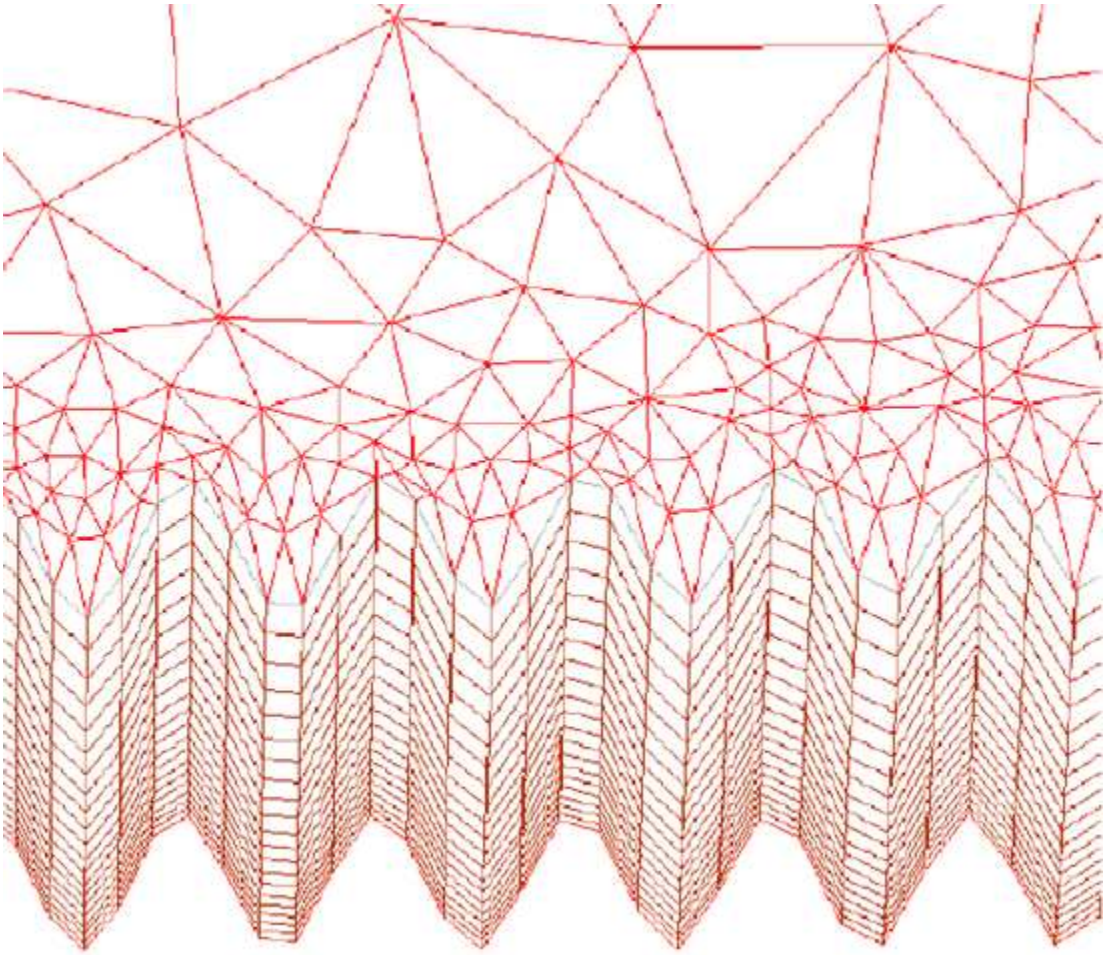
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.26 The Mesh After Applying Boundary Layer Smoothing



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

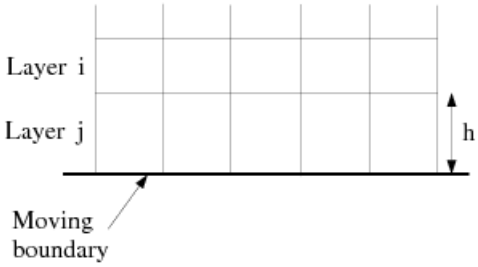
Figure 11.27 Zooming into the Deformed Boundary Layer of the Compliant Strip



11.6.2.2. Dynamic Layering Method

In prismatic (hexahedral and/or wedge) mesh zones, you can use dynamic layering to add or remove layers of cells adjacent to a moving boundary, based on the height of the layer adjacent to the moving surface. The dynamic mesh model in ANSYS FLUENT allows you to specify an ideal layer height on each moving boundary. The layer of cells adjacent to the moving boundary (layer j in *Figure 11.28* (p. 564)) is split or merged with the layer of cells next to it (layer i in *Figure 11.28* (p. 564)) based on the height (h) of the cells in layer j.

Figure 11.28 Dynamic Layering



If the cells in layer j are expanding, the cell heights are allowed to increase until

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$h_{min} > (1+\alpha_s) h_{ideal} \quad \textbf{(11–14)}$$

where h_{min} is the minimum cell height of cell layer j, h_{ideal} is the ideal cell height, and α_s is the layer split factor. Note that ANSYS FLUENT allows you to define h_{ideal} as either a constant value or a value that varies as a function of time or crank angle. When the condition in *Equation 11–14* (p. 565) is met, the cells are split based on the specified layering option. This option can be height based or ratio based.

With the height-based option, the cells are split to create a layer of cells with constant height h_{ideal} and a layer of cells of height $h - h_{ideal}$. With the ratio-based option, the cells are split such that locally, the ratio of the new cell heights is exactly α_s everywhere. *Figure 11.29* (p. 565) and *Figure 11.30* (p. 565) show the result of splitting a layer of cells above a valve geometry using the height-based and ratio-based option.

Figure 11.29 Results of Splitting Layer with the Height-Based Option

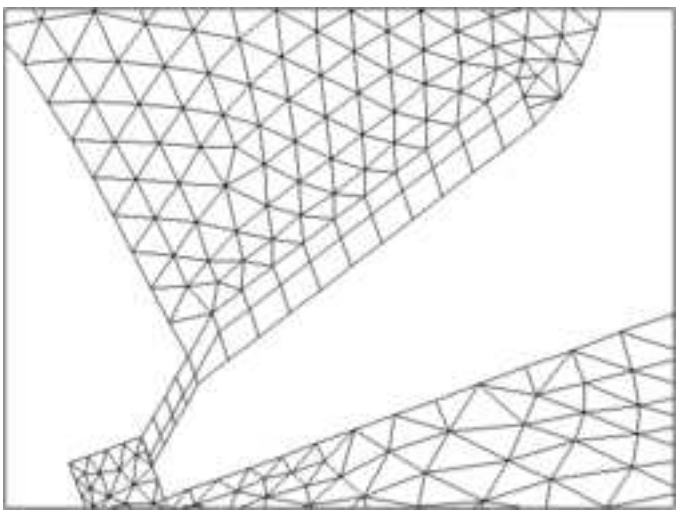
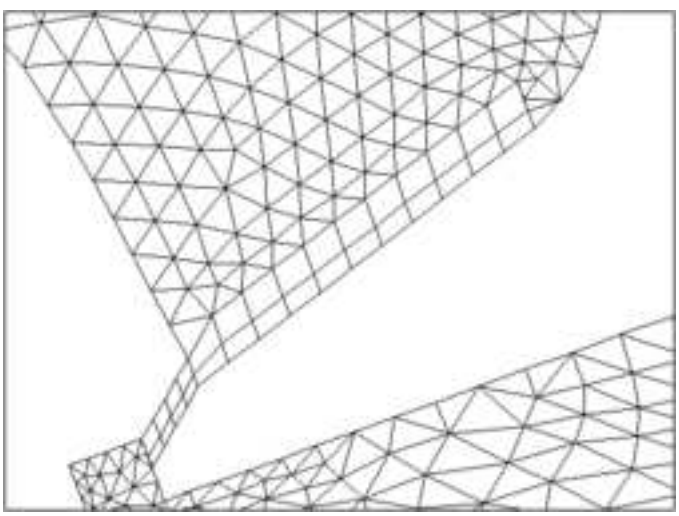


Figure 11.30 Results of Splitting Layer with the Ratio-Based Option



If the cells in layer j are being compressed, they can be compressed until

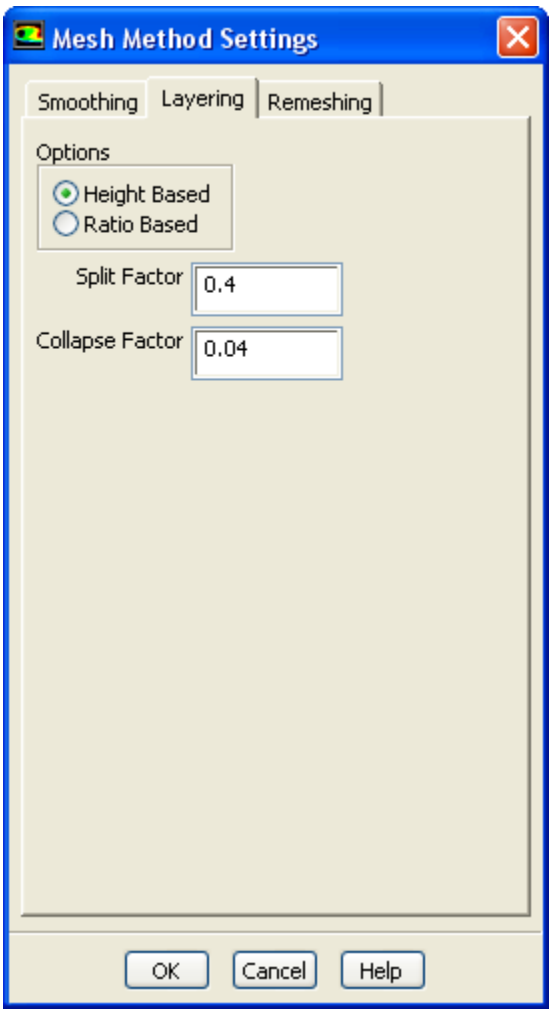
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$h_{min} < \alpha_c h_{ideal} \tag{11–15}$$

where α_c is the layer collapse factor. When this condition is met, the compressed layer of cells is merged into the layer of cells above the compressed layer; i.e., the cells in layer j are merged with those in layer i.

To enable dynamic layering, enable the **Layering** option under **Mesh Methods** in the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.31* (p. 566)). The layering control is specified in the **Layering** tab which can be displayed by clicking **Settings...**.

Figure 11.31 The Layering Tab in the Mesh Method Settings Dialog Box



You can control how a cell layer is split by specifying either **Height Based** or **Ratio Based** under **Options**. Note that for **Height Based**, the height of the cells in a particular new layer will be constant, but you can choose to have this height vary from layer to layer as a function of time or crank angle when you specify the **Cell Height** in the **Dynamic Mesh Zones** dialog box (see *Specifying the Motion of Dynamic Zones* (p. 613) for further details).

The **Split Factor** and **Collapse Factor** (α_s in *Equation 11–14* (p. 565) and α_c in *Equation 11–15* (p. 566), respectively) are the factors that determine when a layer of cells (hexahedra or wedges in 3D, or quadrilaterals in 2D) that is next to a moving boundary is split or merged with the adjacent cell layer, respectively.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.2.2.1. Applicability of the Dynamic Layering Method

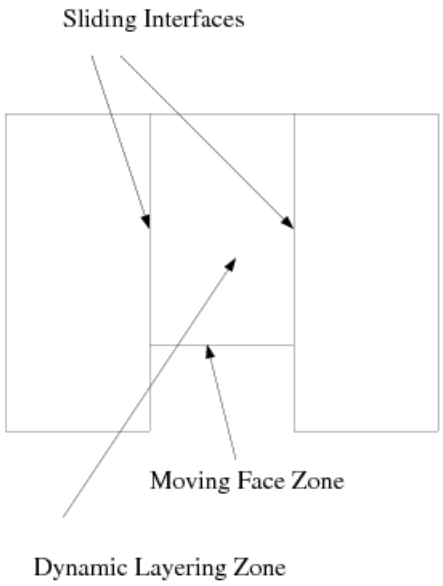
You can use the dynamic layering method to split or merge cells adjacent to any moving boundary provided the following conditions are met:

- All cells adjacent to the moving face zone are either wedges or hexahedra (quadrilaterals in 2D) even though the cell zone may contain mixed cell shapes.
- The cell layers must be completely bounded by one-sided face zones, except when sliding interfaces are used (see *Applicability of the Face Region Remeshing Method* (p. 578)).
- If the bounding face zones are two-sided walls, you must split the wall and wall-shadow pair and use the coupled sliding interface option to couple the two adjacent cell zones.
- Note that you cannot use the dynamic layering method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

If the moving boundary is an internal zone, cells on both sides (possibly with different ideal cell layer heights) of the internal zone are considered for dynamic layering.

If you want to use dynamic layering on cells adjacent to a moving wall that do not span from boundary to boundary, you must separate those cells which are involved in the dynamic layering and use the sliding interfaces capability in ANSYS FLUENT to transition from the deforming cells to the adjacent non-deforming cells (see *Figure 11.32* (p. 567)). For a moving interior face, the zones must be separated such that they are either expanding or collapsing on the same side. No one zone can consist of both expanding and collapsing layers.

Figure 11.32 Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone



11.6.2.3. Remeshing Methods

On zones with a triangular or tetrahedral mesh, the spring-based smoothing method (described in *Spring-Based Smoothing* (p. 552)) is normally used. When the boundary displacement is large compared to the local cell sizes, the cell quality can deteriorate or the cells can become degenerate. This will invalidate the mesh (e.g., result in negative cell volumes) and consequently, will lead to convergence problems when the solution is updated to the next time step.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To circumvent this problem, ANSYS FLUENT agglomerates cells that violate the skewness or size criteria and locally remeshes the agglomerated cells or faces. If the new cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells (with the solution interpolated from the old cells). Otherwise, the new cells are discarded.

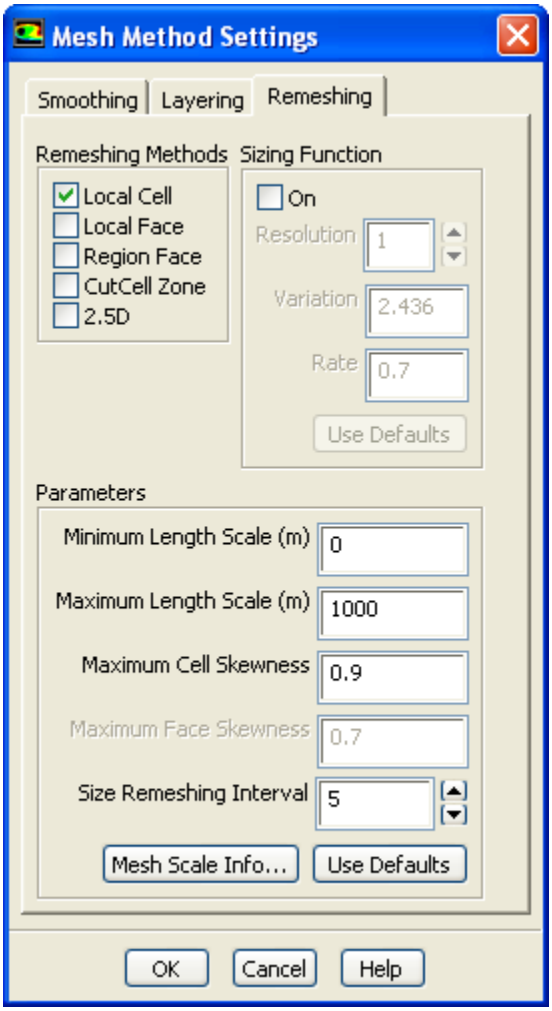
ANSYS FLUENT includes several remeshing methods that include local cell remeshing, zone remeshing, local face remeshing (for 3D flows only), face region remeshing, CutCell zone remeshing (for 3D flows only), and 2.5D surface remeshing (for 3D flows only). Most of the available remeshing methods in ANSYS FLUENT work for triangular-tetrahedral zones and mixed zones where the non-triangular/tetrahedral elements are skipped. The exceptions are the CutCell zone remeshing method and the 2.5D surface remeshing method: the CutCell zone remeshing method works for all cell types (except for polyhedra), and the 2.5D remeshing method only works on hexagonal meshes or wedges extruded from triangular surface elements.

To enable remeshing methods, enable the **Remeshing** option under **Mesh Methods** in the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.13* (p. 550)). Click the **Settings...** button to open the **Mesh Method Settings** dialog box, where you can specify the remeshing method and parameters in the **Remeshing** tab (*Figure 11.33* (p. 569)).

You can view the vital statistics of your mesh by clicking the **Mesh Scale Info...** button at the bottom of the *Mesh Method Settings Dialog Box* (p. 1879). This dialog box displays the *Mesh Scale Info Dialog Box* (p. 1882) where you can view the minimum and maximum length scale values as well as the maximum cell and face skewness values.

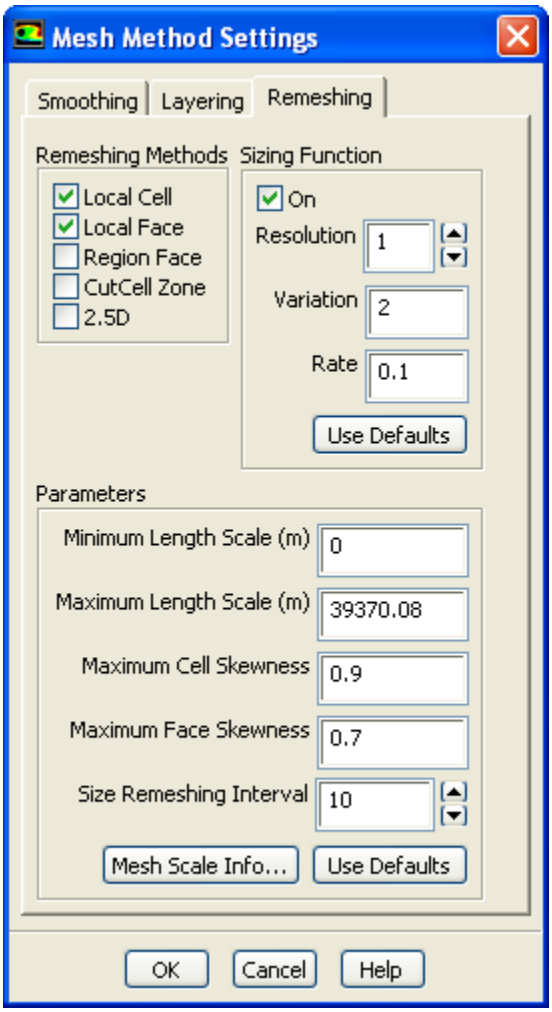
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.33 The Remeshing Tab in the Mesh Method Settings Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.34 The Remeshing Tab in the Mesh Method Settings Dialog Box Using the Sizing Function Option



11.6.2.3.1. Local Remeshing Method

Using the local remeshing method (i.e., local cell remeshing, with or without local face remeshing), ANSYS FLUENT marks cells based on cell skewness and minimum and maximum length scales as well as an optional sizing function.

ANSYS FLUENT evaluates each cell and marks it for remeshing if it meets one or more of the following criteria:

- It has a skewness that is greater than a specified maximum skewness.
- It is smaller than a specified minimum length scale.
- It is larger than a specified maximum length scale.
- Its height does not meet the specified length scale (at moving face zones, e.g., above a moving piston).

11.6.2.3.1.1. Local Cell Remeshing Method

As previously mentioned, in local cell remeshing, ANSYS FLUENT agglomerates cells based on skewness, size, and height (adjacent moving face zones) prior to the movement of the boundary. The size criteria are specified with **Minimum Length Scale** and **Maximum Length Scale**. Cells with length scales below the minimum length scale and above the maximum length scale are marked for remeshing. The value of **Maximum Cell**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skewness indicates the desired skewness of the mesh. By default, the **Maximum Cell Skewness** is set to 0.9 for 3D simulations and 0.6 for 2D simulations. Cells with skewness above the maximum skewness are marked for remeshing.

The marking of cells based on skewness is done at every time step when the local remeshing method is enabled. However, marking based on size and height is performed between the specified **Size Remeshing Interval** since the change in cell size distribution is typically small over one time step.

By default, ANSYS FLUENT replaces the agglomerated cells only if the quality of the remeshed cells has improved.

11.6.2.3.1.1.1. Cell Zone Remeshing Method

If local remeshing of cells within a cell zone is not sufficient, then cell zone remeshing will be used. This method remeshes all cells of a cell zone and the faces of adjacent deforming dynamic face zones. The cell zone remeshing gives the mesher more flexibility to create a new mesh of better quality than the local cell remeshing.

11.6.2.3.1.1.2. Applicability of the Cell Zone Remeshing Method

Zone remeshing has the following limitations:

- Zones with hanging nodes cannot be remeshed.
- In parallel, the zone remeshing will automatically migrate all elements to be remeshed to a single CPU. Consequently, the machine memory will limit the size of the zone that can be remeshed.

11.6.2.3.1.2. Local Face Remeshing Method

The local face remeshing method only applies to 3D geometries. Using this method, ANSYS FLUENT marks the faces (and the adjacent cells) on the deforming boundaries based on the face skewness. This method allows you to remesh locally at deforming boundaries; however, you are not able to remesh across multiple face zones.

Enable the **Local Face** remeshing option and set the **Maximum Face Skewness** to a specific value. In addition, you should enable the **Remeshing** option in the **Meshing Options** tab of the *Dynamic Mesh Zones Dialog Box* (p. 1889) for a deforming zone type (see *Deforming Motion* (p. 619)).

11.6.2.3.1.2.1. Applicability of the Local Face Remeshing Method

If you define deforming face zones in your model and you use local face remeshing in the adjacent cell zone, the faces on the deforming face zone can be remeshed only if the following conditions are met:

- The faces are triangular.
- The faces do not exists across zones or features.
- Note that you cannot use the local face remeshing method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

11.6.2.3.1.3. Local Remeshing Based on Size Functions

Instead of marking cells based on minimum and maximum length scales, ANSYS FLUENT also marks cells based on the size distribution generated by the sizing function if the **Sizing Function** option is enabled.

Cells can be marked using size functions only with the following remeshing methods:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- local cell remeshing
- 2.5D surface remeshing (as described in *2.5D Surface Remeshing Method* (p. 582))

Figure 11.36 (p. 572) demonstrates the advantages of using size functions for local remeshing.

Figure 11.35 Mesh at the End of a Dynamic Mesh Simulation Without Size Functions

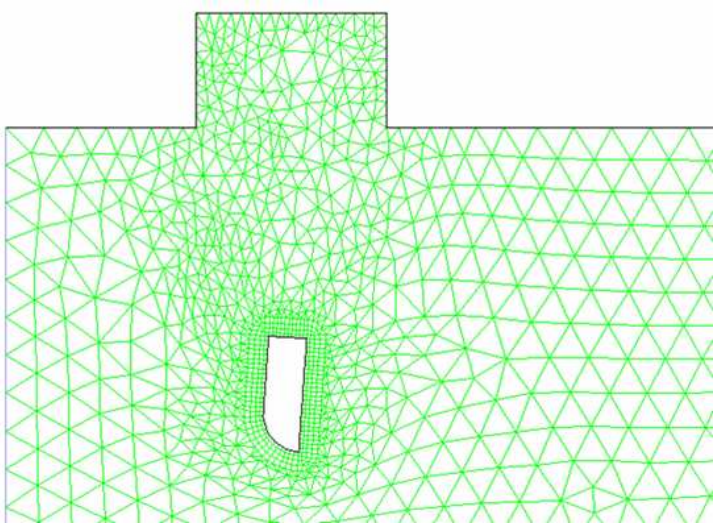
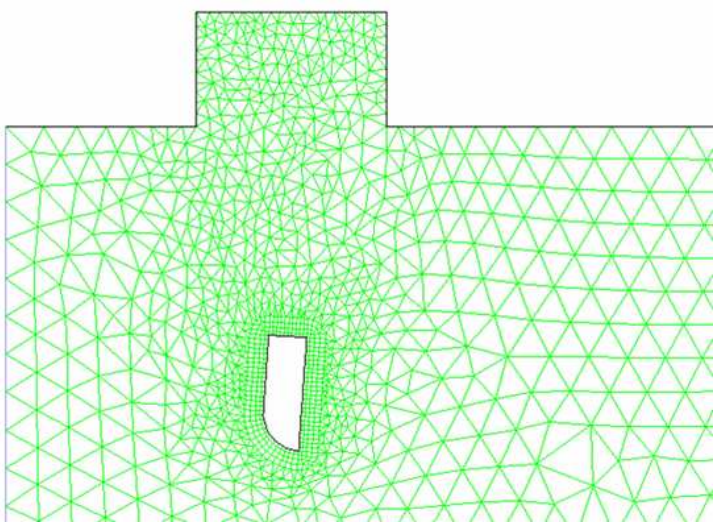


Figure 11.36 Mesh at the End of a Dynamic Mesh Simulation With Size Functions

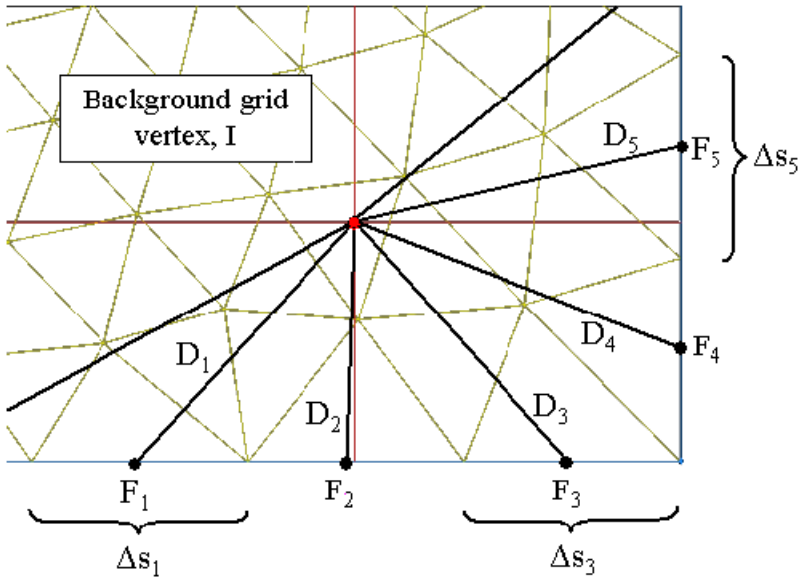


In determining the sizing function, ANSYS FLUENT draws a bounding box around the zone that is approximately twice the size of the zone, and locates the shortest feature length within each fluid zone. ANSYS FLUENT then subdivides the bounding box based on the shortest feature length and the **Size Function Resolution** that you specify. This allows ANSYS FLUENT to create a background mesh.

You control the resolution of the background mesh and a background mesh is created for each fluid zone. The shortest feature length is determined by shrinking a second box around the object, and then selecting the shortest edge on that box. The size function is evaluated at the vertex of each individual background mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.37 Size Function Determination at Background Mesh Vertex I



As seen in *Figure 11.37* (p. 573), the local value of the size function SF_I is defined by

$$SF_I = \left(\frac{\Sigma \frac{1}{D_J} \Delta s_J}{\Sigma \frac{1}{D_J}} \right) \tag{11–16}$$

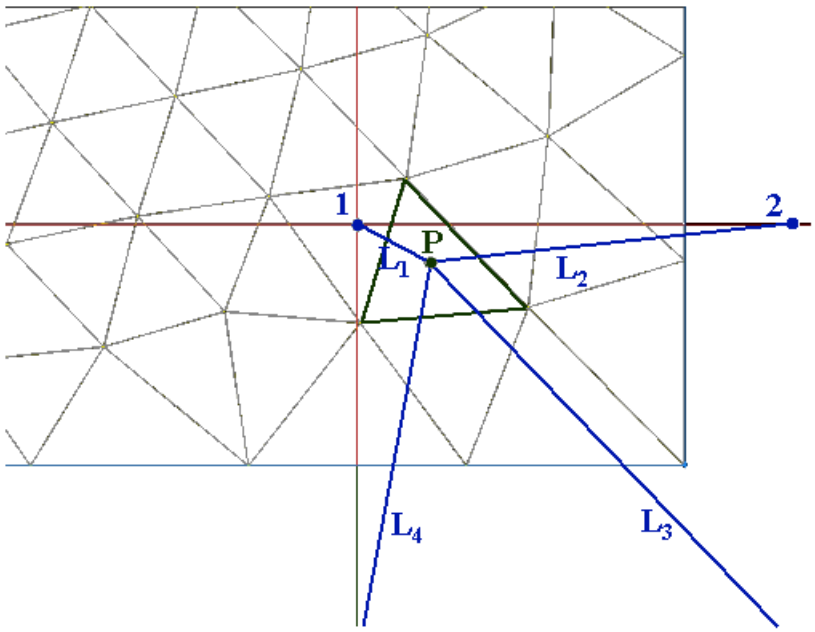
where D_J is the distance from vertex I on the background mesh to the centroid of boundary cell J and Δs_J is the mesh size (length) of boundary cell J.

The size function is then smoothed using Laplacian smoothing. ANSYS FLUENT then interpolates the value of the size function by calculating the distance L_I from a given cell centroid P to the background mesh vertices that surround the cell (see *Figure 11.38* (p. 574)). The intermediate value of the size function $size_b$ at the centroid is computed from

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$size_b = \left(\frac{\Sigma SF_I \frac{1}{L_I}}{\Sigma \frac{1}{L_I}} \right) \tag{11–17}$$

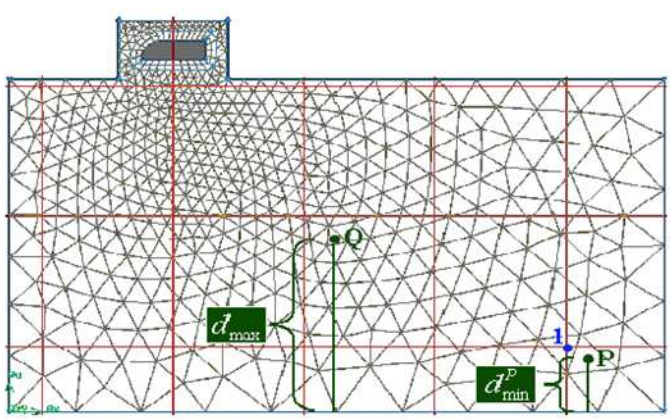
Figure 11.38 Interpolating the Value of the Size Function



Next, a single point Q is located within the domain (see *Figure 11.39* (p. 574)) that has the largest distance d_{max} to the nearest boundary to it. The normalized distance d_b for the given centroid P is given by

$$d_b = \frac{d_{min}^P}{d_{max}} \tag{11–18}$$

Figure 11.39 Determining the Normalized Distance



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the parameters α and β (the **Size Function Variation** and the **Size Function Rate**, respectively), you can write the final value $size_P$ of the size function at point P as

$$size_P = size_b \times \left(1 + \alpha \times d_b^{1+2\beta}\right) = size_b \times \gamma \qquad \textbf{(11–19)}$$

where $size_b$ is the intermediate value of the size function at the cell centroid.

Note that α is the size function *variation*. Positive values mean that the cell size increases as you move away from the boundary. Since the maximum value of d_b is one, the maximum cell size becomes

$$size_{P,max} = size_b \times (1 + \alpha) = size_b \times \gamma_{max} \qquad \textbf{(11–20)}$$

thus, α is really a measure of the maximum cell size.

The factor γ is computed from

$$\gamma = 1 + \alpha d_b^{1+2\beta} \qquad \text{if } \alpha > 0 \qquad \textbf{(11–21)}$$

$$\gamma = 1 + \alpha d_b^{\frac{1}{1-\beta}} \qquad \text{if } \alpha < 0 \qquad \textbf{(11–22)}$$

You can use **Size Function Variation** (or α) to control how large or small an interior cell can be with respect to its closest boundary cell. α ranges from -1 to ∞, an α value of 0.5 indicates that the interior cell size can be, at most, 1.5 the size of the closest boundary cell. Conversely, an α value of -0.5 indicates that the cell size interior of the boundary can be half of that at the closest boundary cell. A value of 0 indicates a constant size distribution away from the boundary.

You can use **Size Function Rate** (or β) to control how rapidly the cell size varies from the boundary. The value of β should be specified such that $-0.99 < \beta < +0.99$. A positive value indicates a slower transition from the boundary to the specified **Size Function Variation** value. Conversely, a negative value indicates a faster transition from the boundary to the **Size Function Variation** value. A value of 0 indicates a linear variation of cell size away from the boundary.

You can also control the resolution of the sizing function with **Size Function Resolution**. The resolution determines the size of the background bins used to evaluate the size distribution with respect to the shortest feature length of the current mesh. By default, the **Size Function Resolution** is 3 in 2D problems, and 1 in 3D problems.

A set of default values (based on the current mesh) is automatically generated if you click **Use Defaults**.

In summary, the sizing function is a distance-weighted average of all mesh sizes on all boundary faces (both stationary and moving boundaries). The sizing function is based on the sizes of the boundary cells, with the size computed from the cell volume by assuming a perfect (equilateral) triangle in 2D and a perfect tetrahedron in 3D. You can control the size distribution by specifying the **Size Function Variation** and the **Size Function Rate**. If you have enabled the **Sizing Function** option, ANSYS FLUENT will agglomerate a cell if

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$size \notin \left[\frac{4}{5}\gamma size_b, \frac{5}{4}\gamma size_b\right] \quad \textbf{(11–23)}$$

where γ is a factor defined by *Equation 11–21* (p. 575) and *Equation 11–22* (p. 575).

Note that the size function is only used for marking cells *before* remeshing. The size function is not used to govern the size of the cell during remeshing.

For steady-state applications (see *Steady-State Dynamic Mesh Applications* (p. 628)), you can instruct ANSYS FLUENT to perform a second round of cell marking and agglomeration after the boundary has moved, based on skewness criteria. The intent is to further improve the mesh quality through additional local remeshing. This optional feature works in conjunction with the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.33* (p. 569)), and operates according to the skewness parameters you set in this dialog box. The size function parameters are not considered during this additional remeshing. Note that enabling this option will increase the time required to update the mesh during the solution.

Important

Additional local remeshing after the boundary has moved is not available for transient dynamic mesh applications, as the resulting numerical method would no longer be conservative.

When you use the **Sizing Function** remeshing option (see *Figure 11.34* (p. 570)), you can control three parameters that govern the size function. You can specify the **Size Function Resolution**, the **Size Function Variation**, and the **Size Function Rate** or you can return to ANSYS FLUENT's default values by using the **Use Defaults** button.

The size function **Resolution** controls the density of the background mesh (see *Local Remeshing Based on Size Functions* (p. 571)). By default, it is equivalent to 3 in 2D simulations and 1 in 3D simulations.

The size function **Variation** corresponds to α in *Equation 11–19* (p. 575). It is the measure of the maximum permissible cell size and it ranges from $-1 < \alpha < +\infty$.

The size function **Rate** corresponds to β in *Equation 11–19* (p. 575). It is the measure of the rate of growth of the cell size, and it ranges from $-0.99 < \beta + + < 0.99$. A value of 0 implies linear growth, whereas higher values imply a slower growth near the boundary with faster growth as one moves toward the interior.

Note

To employ additional local remeshing, first make sure that you have enabled the **Remeshing** option in the *Dynamic Mesh Task Page* (p. 1877).

You can also use the following text command:

`define` → `dynamic-mesh` → `controls` → `remeshing-parameter` → `remeshing-after-moving`?

Finally, type `yes` to the question, `optional remeshing after moving the mesh`?

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.2.3.2. Face Region Remeshing Method

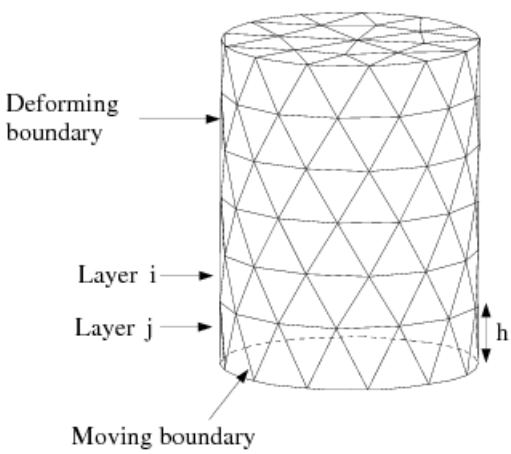
In addition to remeshing the volume mesh, ANSYS FLUENT also allows triangular and linear faces on a deforming boundary to be remeshed. ANSYS FLUENT marks deforming boundary faces for remeshing based on moving and deforming loops of faces.

For face region remeshing, ANSYS FLUENT marks the region of faces on the deforming boundaries at the moving boundary based on minimum and maximum length scales. Once marked, ANSYS FLUENT remeshes the faces and the adjacent cells to produce a very regular mesh on the deforming boundary at the moving boundary (e.g., *Figure 11.40* (p. 577)). Using this method, ANSYS FLUENT is able to remesh across multiple face zones.

For 3D simulations, ANSYS FLUENT allows remeshing with symmetric boundary conditions, and across multiple face zones which includes preserving features not only between the different face zones, but also within a face zone. For more information, see *Feature Detection* (p. 586).

ANSYS FLUENT automatically extracts loops on the boundary of the face zone whose nodes are moving or deforming. Consider a simple tetrahedral mesh of a cylinder whose bottom wall is moving (see *Figure 11.40* (p. 577)). On the deforming boundary, a single loop is generated at the bottom end of the cylinder (where the nodes are moving). ANSYS FLUENT analyzes the height of the faces connected to the nodes on the loop and subsequently, splits or merges the faces depending on the specified maximum or minimum length scale.

Figure 11.40 Remeshing at a Deforming Boundary



If the faces in layer j are expanding, they are allowed to expand until the maximum length scale is reached. Conversely, if the layer is contracting, they are allowed to contract until the minimum length scale is reached. When this condition is met, the compressed layer of faces is merged into the layer of faces above it. The face remeshing is illustrated in *Figure 11.42* (p. 578).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.41 Expanding Cylinder Before Region Face Remeshing

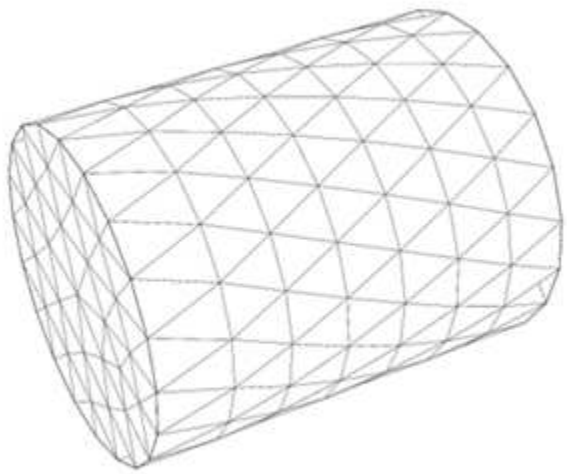
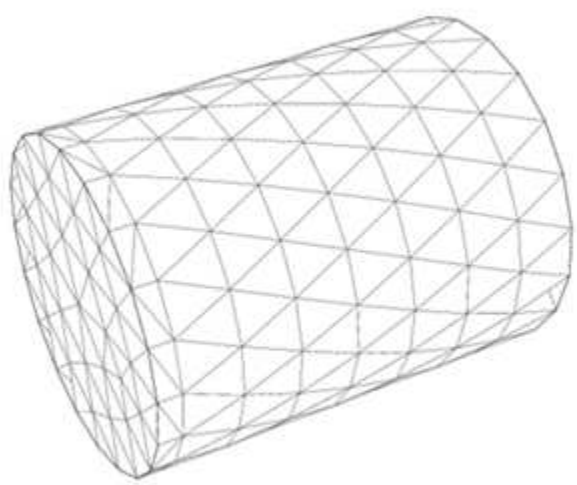


Figure 11.42 Expanding Cylinder After Region Face Remeshing



11.6.2.3.2.1. Applicability of the Face Region Remeshing Method

You can use the face region remeshing method only in cell zones that contain tetrahedral or triangular cells.

If you define deforming face zones in your model and you use face region remeshing in the adjacent cell zone, the faces on the deforming face zone can be remeshed only if the following conditions are met:

- The faces are triangular (or linear in 2D).

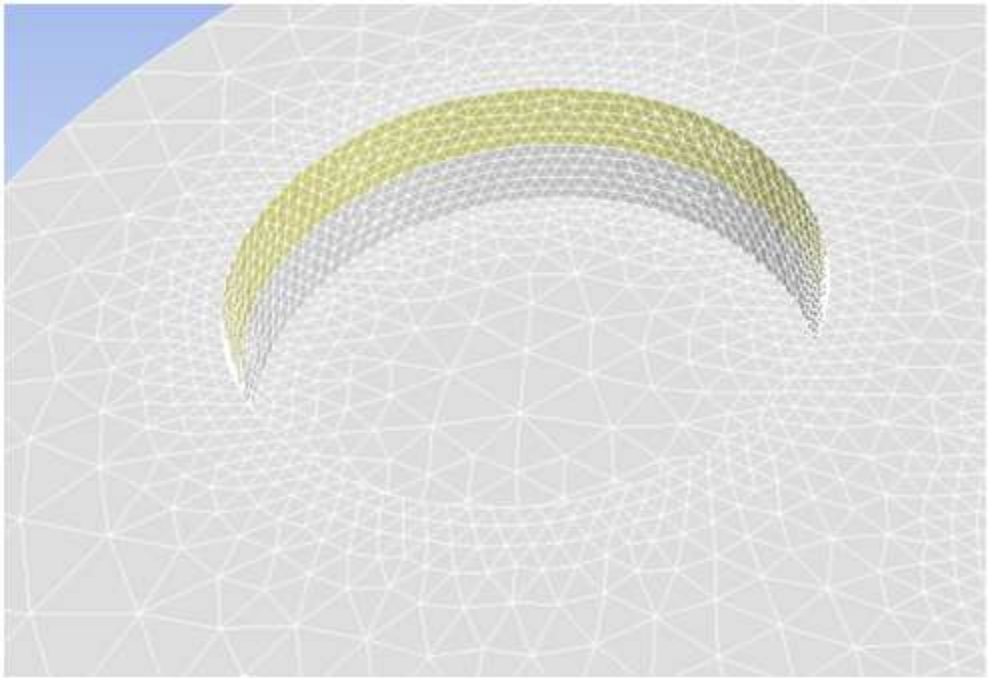
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Note that you cannot use the face region remeshing method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

11.6.2.3.3. CutCell Zone Remeshing Method

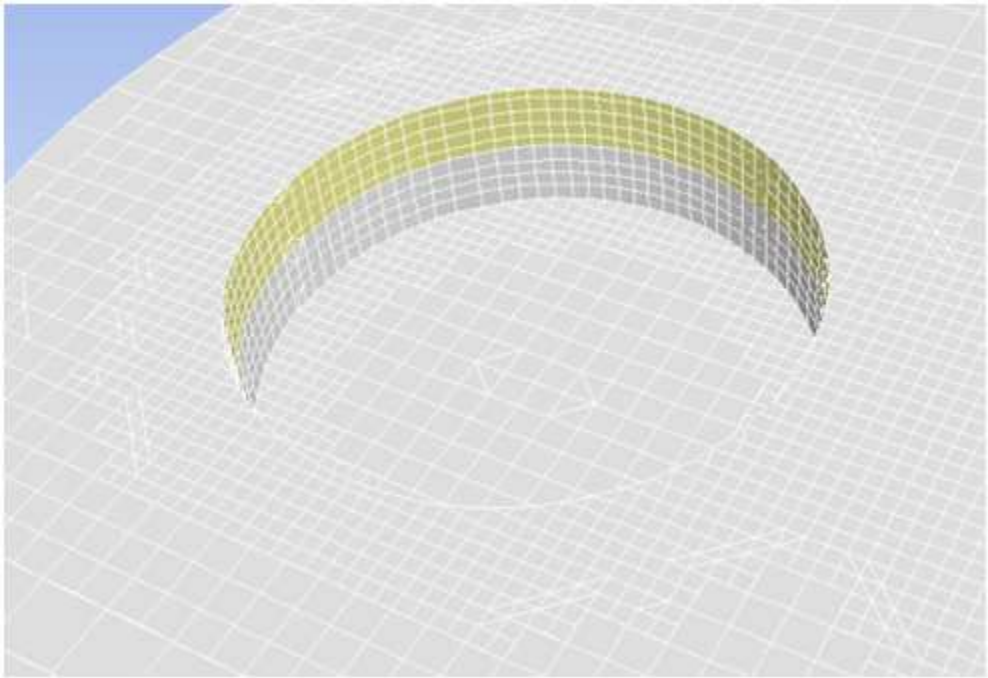
The CutCell zone remeshing method is available to remesh a complete cell zone (including all boundary zones of the remeshed cell thread). This method is available for 3D simulations only. The existing volume mesh is replaced by a predominantly Cartesian mesh. When used as part of a transient simulation, the remeshing occurs at predefined intervals and whenever the mesh quality in the cell zone is deemed poor. Examples of meshes before and after CutCell zone remeshing are shown in *Figure 11.43* (p. 579) and *Figure 11.44* (p. 580), respectively.

Figure 11.43 Unstructured Tetrahedral Mesh Before CutCell Zone Remeshing



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.44 Mesh After CutCell Zone Remeshing



During CutCell zone remeshing, a uniform Cartesian grid is locally refined using size functions, in order to resolve the initial mesh with sufficient accuracy. The resulting mesh consists mostly of hexahedral elements, which reduces the cell count compared to unstructured tetrahedral meshing. Additional element types are used near the boundaries to closely resolve complex shapes. The remeshing takes place at regular intervals, or whenever the mesh quality deteriorates due to mesh motion.

The CutCell zone remeshing method not only replaces the volume mesh, it also replaces the complete surface mesh of the remeshed cell zone. Consequently, this method can only be used to remesh cell zones which are either stand-alone zones or are only connected to other cell zones through non-conformal interfaces.

The **CutCell Zone** remeshing method option is available in the **Remeshing Methods** group box. Note that the **CutCell Zone** remeshing method does not use any of the parameters specified in the **Mesh Method Settings** dialog box. The parameters used to control **CutCell Zone** remeshing are specified in the **Dynamic Mesh Zones** dialog box when setting up the zone and boundaries.

For more information about CutCell meshes, see the TGrid User's Guide.

11.6.2.3.3.1. Applicability of the CutCell Zone Remeshing Method

The CutCell zone remeshing method can be applied to cell zones, with the following limitations:

- The case must be 3D.
- The cell zone cannot contain polyhedral elements.
- The cell zone cannot be conformally connected to other cell zones; that is, a CutCell zone needs to be a stand-alone cell zone, or can only be connected to another zone through a non-conformal interface. In the latter case, the non-conformal interface is cleared before the remeshing and automatically recreated after the remeshing.

Note that the CutCell zone remeshing method can be applied to cell zones that contain adapted cells.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In parallel, all cells of the remeshed zone will be automatically migrated to and remeshed on a single CPU. Consequently, the machine memory will limit the size of the zone that can be remeshed. The mesh is automatically repartitioned after the remeshing.

In addition to the limitations listed previously, you should also note the following with regard to CutCell zone remeshing:

- Internal coupled walls, such as baffles, are discarded during the remeshing. The method is designed to primarily remesh cell zones with a single interior zone.
- Interior jump boundary condition zones (e.g., fans, porous jumps), are discarded during the remeshing.
- Conformal periodic boundary conditions are not maintained after the remeshing. Any existing conformal periodic boundaries on the zone being remeshed should slit and replaced with non-conformal periodic boundaries.
- If the zone being remeshed contains multiple interior zones prior to the remeshing, all interior zones will be collected into a single interior zone during the CutCell zone remeshing.

11.6.2.3.3.2. Using the CutCell Zone Remeshing Method

In order to apply CutCell zone remeshing to a cell zone, begin by enabling the **CutCell Zone** option in the **Remeshing Methods** group box in the **Remeshing** tab of the **Mesh Method Settings** dialog box (see *Figure 11.33* (p. 569)). Next, use the **Dynamic Mesh Zones** dialog box to create a deforming dynamic zone for the cell zone (see *Deforming Motion* (p. 619)). Then enable **CutCell** in the **Remeshing Option** group box and enter the relevant remeshing parameters, including: the maximum mesh size for the Cartesian cells; the global size function growth rate; and the minimum orthogonal quality and remeshing interval to control the remeshing frequency.

ANSYS FLUENT allows you to specify either a soft or a mesh-based size function to control how the Cartesian mesh increases in size from the boundary toward the interior of the cell zone. For the soft size function, you specify a maximum mesh size for each boundary zone of the CutCell cell zone. When you use the mesh-based size function, ANSYS FLUENT analyzes the existing mesh to evaluate the necessary mesh refinement at the boundary. See step 4. of *Stationary Zones* (p. 614) for more information about these size functions. You have control over the size function types and parameters by defining the boundary zones as dynamic zones. If no boundary zone adjacent to the CutCell zone is defined as a dynamic zone, ANSYS FLUENT will automatically apply mesh-based size functions to each CutCell boundary zone and use the global growth rate entered for the cell zone to control the remeshing.

Note that the size functions used by the CutCell zone remeshing method are unrelated to the size function used for local cell remeshing (as described in *Local Remeshing Based on Size Functions* (p. 571)).

It is recommended that you first manually apply the CutCell zone remeshing method (as described in the section that follows) before using it as part of a transient simulation, as this allows you to evaluate if your remeshing parameters are suitable before running a calculation that is computationally expensive.

11.6.2.3.3.3. Applying the CutCell Zone Remeshing Method Manually

Although the CutCell zone remeshing method is primarily intended as an automatic remeshing method during dynamic mesh updates, you can also manually create a CutCell mesh (i.e., you can remesh without running a calculation) by using the following text command:

`define` → `dynamic-mesh` → `actions` → `remesh-cell-zone-cutcell`

You will be prompted to enter the global parameters for the CutCell zone remeshing. If no dynamic zones are set up in your case, ANSYS FLUENT will use the global parameters you enter and apply mesh-based size

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

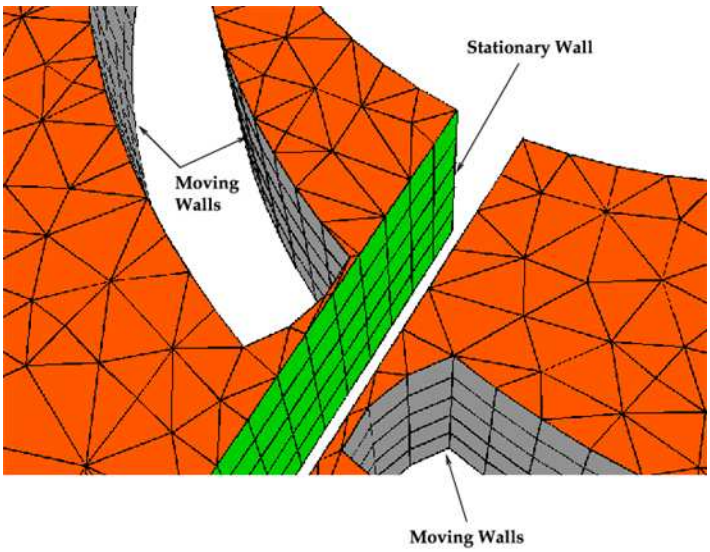
functions to each boundary of the CutCell zone. If the case is set up as a dynamic mesh case, manual remeshing will use the information specified on each CutCell boundary zone. During manual CutCell zone remeshing, the global parameters are taken from the text interface input, and the global parameters specified on the CutCell cell zone are ignored.

When executing manual remeshing through the `remesh-cell-zone-cutcell` text command, you will be prompted to specify whether topology warnings should be ignored. If you do not ignore the topology warnings, ANSYS FLUENT will analyze the zone being remeshed and report any interior face zones that will get discarded during the remeshing. If the zone being remeshed has periodic boundary zones, you will also be prompted to either abort the remeshing process or slit the periodics in order to proceed.

11.6.2.3.4. 2.5D Surface Remeshing Method

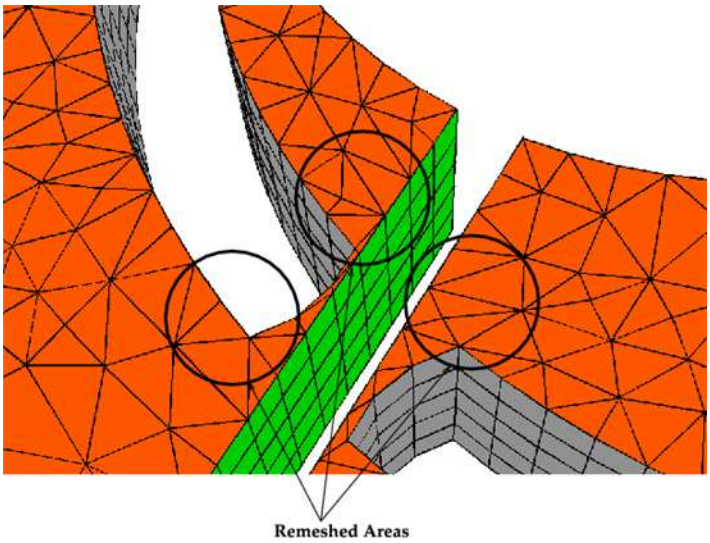
The 2.5D surface remeshing method only applies to extruded 3D geometries and is similar to local remeshing in two dimensions on a triangular surface mesh (not a mixed zone). Faces on a deforming boundary are marked for remeshing based on face skewness, minimum and maximum length scale; the 2.5D remeshing method also gives you the option of marking cells using size functions, as described in *Local Remeshing Based on Size Functions* (p. 571).

Figure 11.45 Close-Up of 2.5D Extruded Flow Meter Pump Geometry Before Remeshing and Laplacian Smoothing



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.46 Close-Up of 2.5D Extruded Flow Meter Pump Geometry After Remeshing and Laplacian Smoothing



11.6.2.3.4.1. Applicability of the 2.5D Surface Remeshing Method

The following applies to the 2.5D surface remeshing method:

- Triangular faces get remeshed based on marking.
- Extruded prisms get remeshed based on the remeshing of the triangular face. Only extruded regions get remeshed, not mixed regions.
- Note that you cannot use the 2.5D surface remeshing method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.
- In the extruded/coopered mesh, area zone changes are not allowed. In such cases, make sure that the face zones at the extruded mesh area do not change from the top to the bottom. For more information about the 2.5D model, see *Using the 2.5D Model* (p. 583).
- Periodics are not supported at the extruded zones.

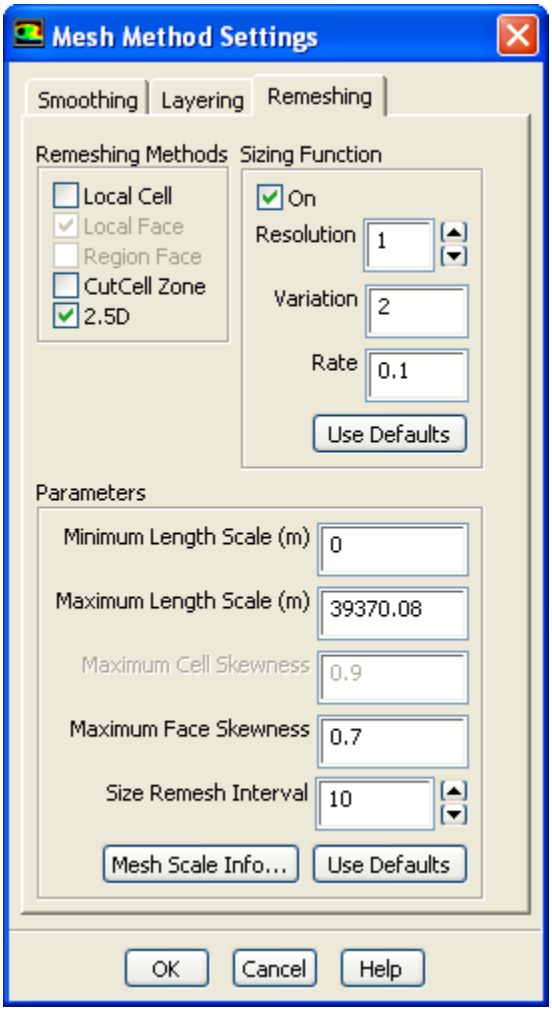
11.6.2.3.4.2. Using the 2.5D Model

For 3D simulations only, you can select the **2.5D** model under the **Remeshing** tab in the *Mesh Method Settings Dialog Box* (p. 1879). This model allows for a specific subset of remeshing techniques.

The 2.5D mesh essentially is a 2D triangular mesh which is expanded, or extruded, along the normal axis of the specific dynamic zone that you are interested in modeling. The triangular surface mesh is remeshed and smoothed on one side, and the changes are then extruded to the opposite side. Rigid body motion is applied to the moving face zones, while the triangular extrusion surface is assigned to a deforming zone with remeshing and smoothing enabled. The opposite side of the triangular mesh is assigned to be a deforming zone as well, with only smoothing enabled, as in *Figure 11.48* (p. 585).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.47 The Remeshing Tab for the 2.5D Model

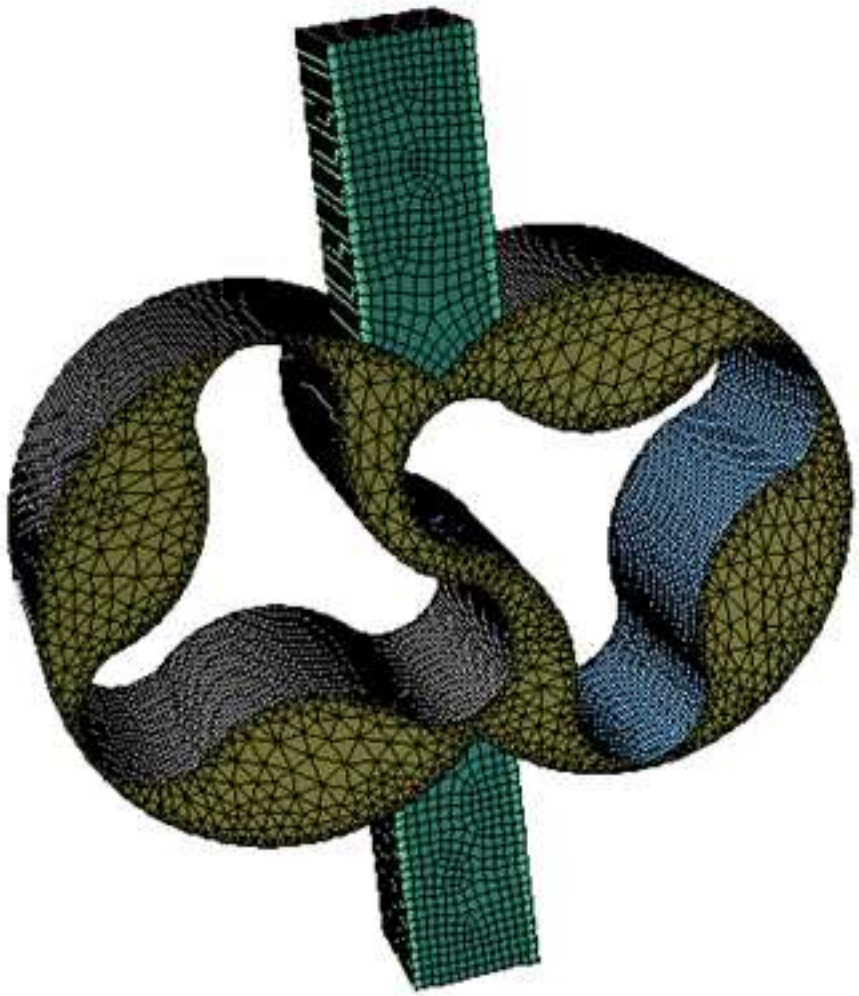


For more information on setting smoothing and remeshing parameters, see *Dynamic Mesh Update Methods* (p. 551).

The 2.5D model only applies to mappable (i.e., extrudable) mesh geometries such as pumps, as in *Figure 11.48* (p. 585). Only the aspects of the geometry that represent the "moving parts" need to be extruded in the mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.48 2.5D Extruded Gear Pump Geometry



Important

You must only apply smoothing to the opposite side of the extruded mesh, since ANSYS FLUENT requires the geometry information for the dynamic zone. ANSYS FLUENT projects the nodes back to its geometry after the extrusion. Without this geometry information, the dynamic zones tends to lose its integrity.

Important

In parallel, a partition method that partitions perpendicular to the extrusion surface should be used. For example, if the normal of the extrusion surface points in the x-direction then Cartesian-Y or Cartesian-Z would be the perfect partition methods.

The 2.5D model is used in combination with a `DEFINE_GRID_MOTION` UDF. (See Hooking DEFINE_GRID_MOTION UDFs in the UDF Manual for information about hooking this UDF.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This UDF is associated with the extrusion surface that is adjacent to the cell zone, in turn applying the same deformation to the entire cell zone. This approach is particularly useful when modeling gear pumps that are predominantly extruded hexahedral meshes. For more information about this UDF, contact your support engineer.

11.6.2.3.5. Feature Detection

For 3D simulations, ANSYS FLUENT allows you to preserve features on deforming zones not only between the different face zones, but also within a face zone.

In the **Geometry Definition** tab of the *Dynamic Mesh Zones Dialog Box* (p. 1889), for any geometry definition, you can indicate whether you want to include features of a specific angle by selecting **Include Features** under **Feature Detection** and setting the **Feature Angle** (the zonal feature angle α) in degrees. If the angle β between adjacent faces is bigger than the specified angle, then the feature is recognized (i.e., $\cos(\beta)\cos(\alpha)$).

11.6.2.3.5.1. Applicability of Feature Detection

The following items are applicable for use with feature detection:

- Feature remeshing is only possible with face region remeshing.
- Features are preserved by local face remeshing, i.e., there is no local face remeshing across features.
- Smoothing methods preserve features, i.e., nodes at feature edges are not allowed to be smoothed.

11.6.2.4. Volume Mesh Update Procedure

The volume mesh is updated automatically based on the methods described in *Dynamic Mesh Update Methods* (p. 551). ANSYS FLUENT decides which method to use for a particular zone based on which model is enabled and the shape of the cells in the zone. For example, if the boundaries of a tetrahedral cell zone are moving, and unless the zone has been set up for CutCell zone remeshing, the spring-based or diffusion-based smoothing and local remeshing methods will be used to update the volume mesh in this zone. If the zone consists of prismatic (hexahedral and/or wedge) cells, then the dynamic layer method will be used to determine where and when to insert and remove cell layers. On extruded prism zones, the 2.5D surface meshing method will be used.

Depending on which model is enabled, ANSYS FLUENT automatically determines which method to use by visiting the adjacent cell zones and setting appropriate flags for the volume mesh update methods to be used. If you specify the motion for a cell zone, ANSYS FLUENT will visit all of the neighboring cell zones and set the flags appropriately. If you specify the motion of a boundary zone, ANSYS FLUENT will analyze only the adjacent cell zones. If a cell zone does not have any moving boundaries, then no volume mesh update method will be applied to the zone.

Important

Note that as a result of the remeshing procedures, updated meshes may be slightly different when dynamic meshes are used in parallel ANSYS FLUENT, and therefore very small differences may arise in the solutions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

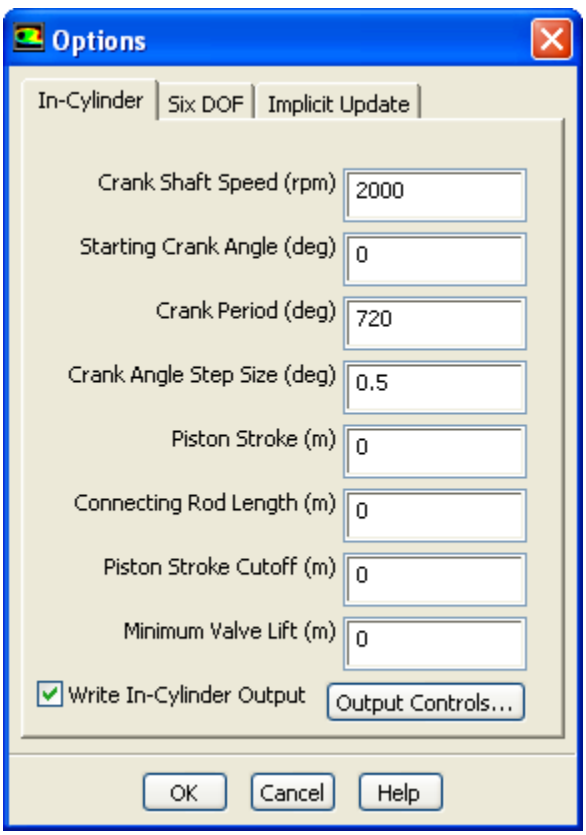
Important

Note that if your dynamic mesh model consists of numerous shell conduction zones, the mesh update may be very time consuming because all shells are deleted and recreated during the mesh update.

11.6.3. In-Cylinder Settings

You can enable the **In-Cylinder** option in the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.13* (p. 550)) for transient problems. Then click the **Settings...** button in the **Options** group box to open the *Options Dialog Box* (p. 1883). Click the **In-Cylinder** tab and specify the **Crank Shaft Speed**, the **Starting Crank Angle**, and the **Crank Period**, which are used to convert between flow time and crank angle. You must also specify the time step to use for advancing the solution in terms of crank angle in **Crank Angle Step Size**. By default, ANSYS FLUENT assumes a **Crank Angle Step Size** of 0.5 degree.

Figure 11.49 The In-Cylinder Tab of the Options Dialog Box



ANSYS FLUENT provides a built-in function to calculate the piston location as a function of crank angle. If the piston motion is specified using this function, you need to specify the **Piston Stroke** and **Connecting Rod Length**. The piston location is calculated using

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$p_s = L + \frac{A}{2}(1 - \cos(\theta_c)) - \sqrt{L^2 - \frac{A^2}{4}\sin^2(\theta_c)} \quad \textbf{(11–24)}$$

where p_s is the piston location (0 at top-dead-center (TDC) and A at bottom-dead-center (BDC)), L is the connecting rod length, A is the piston stroke, and θ_c is the current crank angle. The current crank angle is calculated from

$$\theta_c = \theta_s + t\Omega_{shaft} \quad \textbf{(11–25)}$$

where θ_s is the **Starting Crank Angle** and Ω_{shaft} is the **Crank Shaft Speed**.

The **Piston Stroke Cutoff** and **Minimum Valve Lift** values are used to control the actual values of the valve lift and piston stroke such that

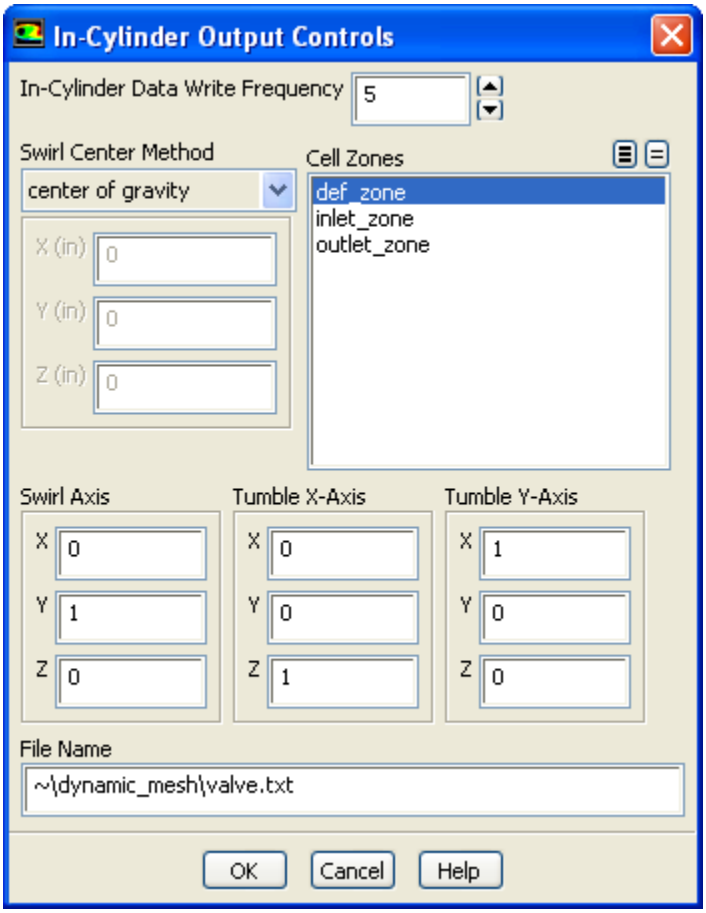
$$\begin{aligned} v_{lift} &= \max\left(v_{lift}^{c}, v_{lift}^{min}\right) \\ p_s &= \min\left(p_s^{c}, p_s^{min}\right) \end{aligned} \quad \textbf{(11–26)}$$

where v_{lift}^{c} is the valve lift computed from the appropriate valve profiles, v_{lift}^{min} is the **Minimum Valve Lift**, p_s^{c} is the stroke calculated from *Equation 11–24* (p. 588), and p_s^{min} is the **Piston Stroke Cutoff**. (See *Defining Motion/Geometry Attributes of Mesh Zones* (p. 594) on how the **Piston Stroke Cutoff** is used to control the onset of layering in the cylinder chamber.)

Enable the **Write In-Cylinder Output** option then click the **Output Controls...** button if you want to specify specific output parameters. The *In-Cylinder Output Controls Dialog Box* (p. 1884) will open, where you can specify various quantities needed for the calculation of swirl and tumble along with the frequency of writing the output to the chosen file. Swirl is used to describe circulation about the cylinder axis. Tumble flow circulates around an axis perpendicular to the cylinder axis, orthogonal to swirl flow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.50 The In-Cylinder Output Controls Dialog Box



The following list describes the *In-Cylinder Output Controls Dialog Box* (p. 1884).

In-Cylinder Data Write Frequency

is an integer entry specifying the interval in number of time-steps. Make sure that a value other than 0 is used for the frequency, in order to allow you to complete your setup.

Swirl Center Method

is a drop-down list which allows you to select the method to calculate the swirl center. The list contains **center of gravity** and **fixed**, with **center of gravity** being the default value.

center of gravity

option calculates the swirl center inside the code and is used as the center of gravity of the chosen cell zones.

fixed

option enables you to specify a swirl center in the entries below the drop-down list.

In addition to these two options, you can chose to use your own compiled UDF to calculate the swirl center.

For details on using a dynamic mesh UDF, see the UDF Manual for information on user-defined functions.

Cell Zones

is a list which displays the names of all existing cell zones in the case files. You can select only the zones relevant for the swirl and tumble calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Swirl Axis
specifies the swirl axis with three entries for the directional components. By default, **X**, **Y**, **Z** = **0**, **1**, **0**.

Tumble X-Axis
specifies the directional components of **Tumble X-Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **0**, **0**, **1**. This applies only in 3D.

Tumble Y-Axis
specifies the directional components of **Tumble Y-Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **1**, **0**, **0**. This applies only in 3D.

File Name
specifies the name of the **In-Cylinder** output file. By default, the file name contains the name of the case file appended with a `.txt` extension.

The **In-Cylinder** specific output controls can also be controlled using the TUI as follows:

```
 Go to
 define/dynamic-mesh/controls/in-cylinder-output?
Enable in-cylinder output?[no] yes
Output Write Frequency[0] 10
Cell zone name/id(1)[()] 2
Cell zone name/id(1)[()]
File Name[''/nfs/devvault/data9/ic-sp-output.txt'']
Swirl Center Method: (fixed cg user-defined)
 Option[cg]
 Swirl Axis x[0]
 Swirl Axis y[1]
 Swirl Axis z[0]
 Tumble X-Axis x[0]
 Tumble X-Axis y[0]
 Tumble X-Axis z[1]
 Tumble Y-Axis x[1]
 Tumble Y-Axis y[0]
 Tumble Y-Axis z[0]
```

If you select **fixed** as the choice at **Swirl Center Method** then you will be prompted to enter the swirl center as follows:

```
Swirl Center(x) (mm) [0]
Swirl Center(y) (mm) [0]
Swirl Center(z) (mm) [0]
```

If a swirl center method UDF has been compiled already and loaded into UDF then you can choose **user-defined** as the swirl center method option, in such a case the following is the sequence of prompts.

```
Swirl Center UDF[] swirl_udf::libudf
```

If the name of the UDF library is **libudf** then you can omit this and enter in the swirl center `UDF[]swirl_udf`, otherwise the name of the UDF followed by the UDF library name with symbol:: in between, should be entered.

By filling up the various entries that are needed in the *In-Cylinder Output Controls Dialog Box* (p. 1884) and pressing the **OK** button, the swirl and tumble calculations will be written at the chosen frequency to the chosen file while doing the solution run. Details of the quantities written to the file are as follows:

CA = Crank Angle

m = Mass of the entire fluid contained in the selected cell zones

L = Angular momentum vector of fluid mass contained in selected cell zones with respect to the swirl center

$|\vec{L}|$ = Magnitude of angular momentum of fluid

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\vec{sa}$ = Swirl Axis

$\vec{tx}$ = Tumble X-Axis

$\vec{ty}$ = Tumble Y-Axis

$\mathbf{I}_{sa}$ = Moment of inertia of the fluid mass about Swirl axis

$\mathbf{I}_{tx}$ = Moment of inertia of the fluid mass about Tumble X-Axis

$\mathbf{I}_{ty}$ = Moment of inertia of the fluid mass about Tumble Y-Axis

· = Dot product between two vectors

Altogether, the previous quantities are combined to yield eight columns of data in the output file, as shown in the figure that follows:

Figure 11.51 Sample Output File Showing Various Quantities

CA	(L . sa)	(L . tx)	(L . ty)	|L|	I_{sa}	I_{tx}	I_{ty}
350.00	0.0000e+00	0.0000e+00	0.0000e+00	0.0000e+00	1.1474e-07	7.0881e-08	6.8234e-08
365.00	6.4910e-07	7.6332e-07	-2.3923e-08	1.0023e-06	9.9355e-08	6.1951e-08	5.9120e-08
380.00	-1.6684e-06	-2.3585e-06	-7.8704e-08	2.8900e-06	1.5460e-07	9.2601e-08	9.2951e-08
395.00	-3.0125e-05	8.1780e-06	2.9730e-06	3.1357e-05	2.9557e-07	1.7709e-07	1.8031e-07
410.00	-8.9259e-05	1.7637e-05	-1.3191e-05	9.1936e-05	4.9396e-07	3.1077e-07	3.1660e-07
425.00	-2.1336e-04	2.4657e-05	-4.6908e-05	2.1984e-04	7.3845e-07	5.1056e-07	5.1811e-07
440.00	-3.9555e-04	6.6114e-05	-1.0156e-04	4.1370e-04	1.0084e-06	7.8920e-07	7.9931e-07
455.00	-6.0621e-04	1.2651e-04	-1.7990e-04	6.4487e-04	1.2833e-06	1.1499e-06	1.1623e-06
470.00	-8.1472e-04	2.0109e-04	-2.6251e-04	8.7927e-04	1.5486e-06	1.5784e-06	1.5913e-06
485.00	-9.9456e-04	2.7342e-04	-3.6243e-04	1.0933e-03	1.7921e-06	2.0409e-06	2.0526e-06
500.00	-1.1160e-03	2.9711e-04	-4.5477e-04	1.2412e-03	2.0003e-06	2.4850e-06	2.4945e-06

11.6.3.1. Using the In-Cylinder Option

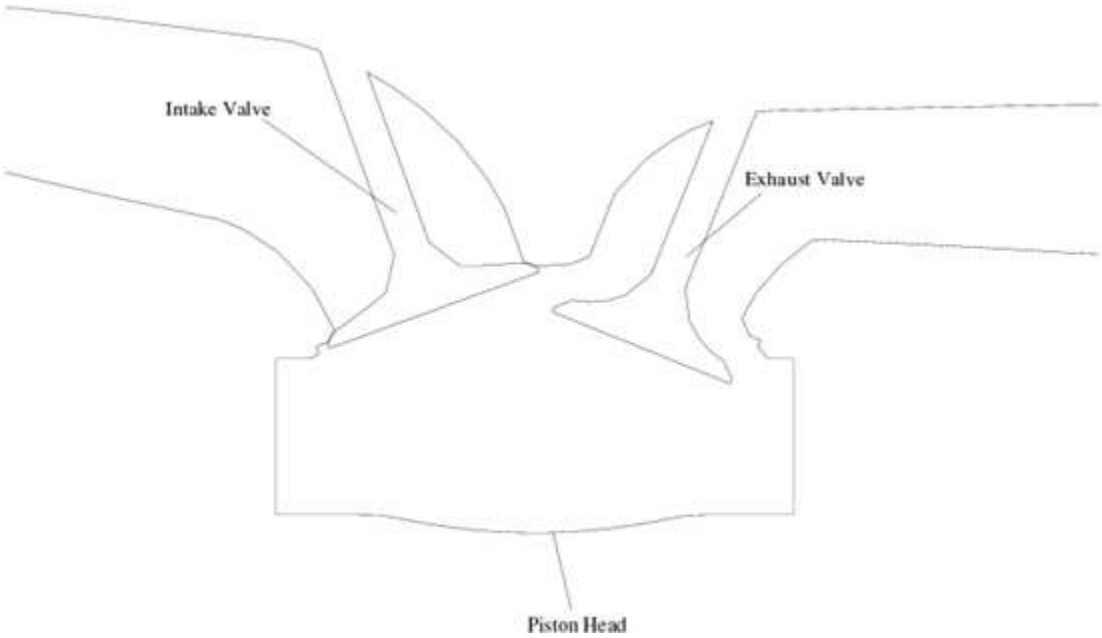
This section describes the problem setup procedure for an in-cylinder dynamic mesh simulation.

11.6.3.1.1. Overview

Consider the 2D in-cylinder example shown in *Figure 11.52* (p. 592) for a typical pent-roof engine.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.52 A 2D In-Cylinder Geometry



In setting up the dynamic mesh model for an in-cylinder problem, you need to consider the following issues:

- how to provide the proper mesh topology for the volume mesh update methods (smoothing, dynamic layering, and local or zonal remeshing)
- how to define the motion attributes and geometry for the valve and piston surfaces
- how to address the opening and closing of the intake and exhaust valves
- how to specify the sequence of events that controls the in-cylinder simulation

11.6.3.1.2. Defining the Mesh Topology

ANSYS FLUENT requires that you provide an initial volume mesh with the appropriate mesh topology such that the various mesh update methods described in *Dynamic Mesh Update Methods* (p. 551) can be used to automatically update the dynamic mesh. However, ANSYS FLUENT does not require you to set up all in-cylinder problems using the same mesh topology. When you generate the mesh for your in-cylinder problem, you need to consider the various mesh regions that you can identify as moving, deforming, or stationary, and generate these mesh regions with the appropriate cell shape.

The mesh topology for the example problem in *Figure 11.52* (p. 592) is shown in *Figure 11.53* (p. 593), and the corresponding volume mesh is shown in *Figure 11.54* (p. 593).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.53 Mesh Topology Showing the Various Mesh Regions

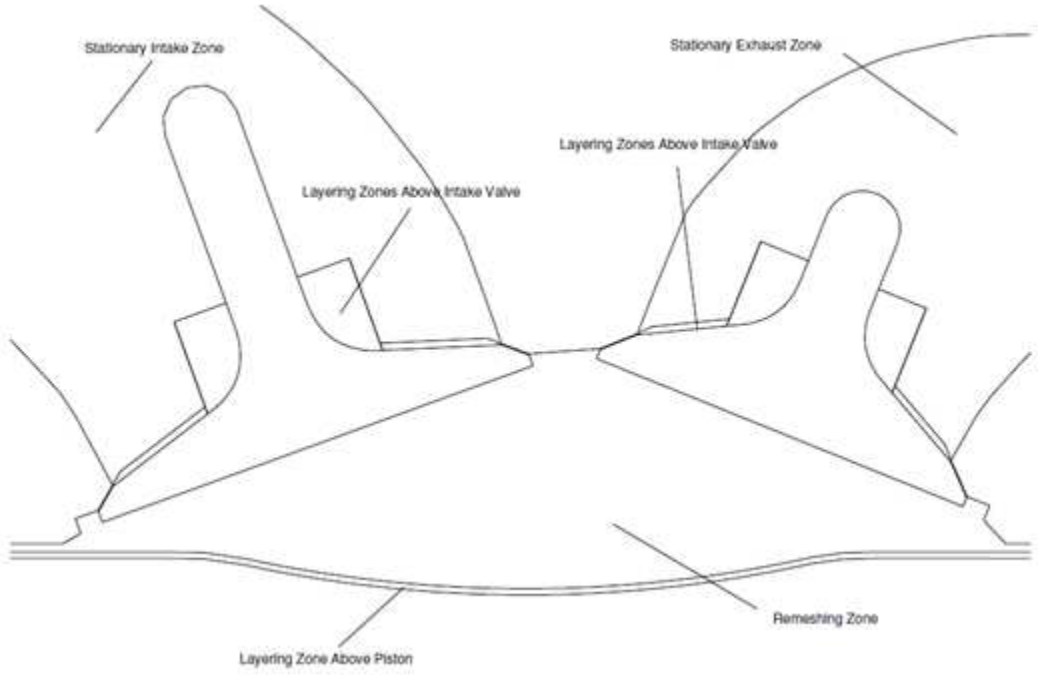
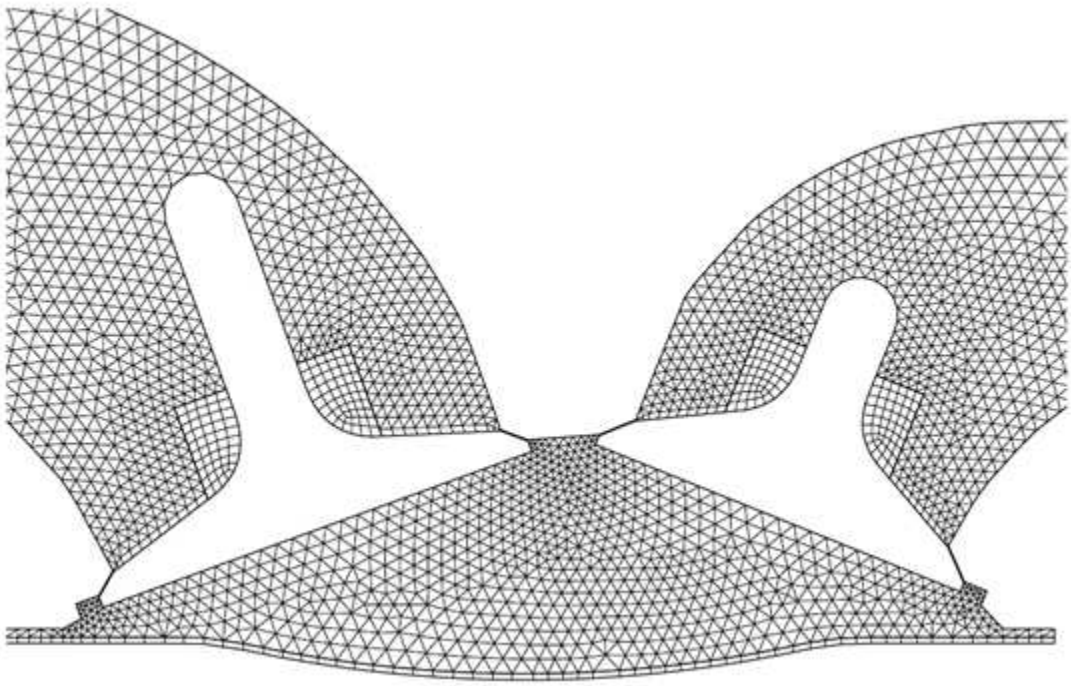


Figure 11.54 Mesh Associated With the Chosen Topology



Because of the rectilinear motion of the moving surfaces, you can use dynamic layering zones to represent the mesh regions swept out by the moving surfaces. These regions are the regions above the top surfaces of the intake and exhaust valves and above the piston head surface, and must be meshed with quadrilateral or hexahedral cells (as required by the dynamic layering method).

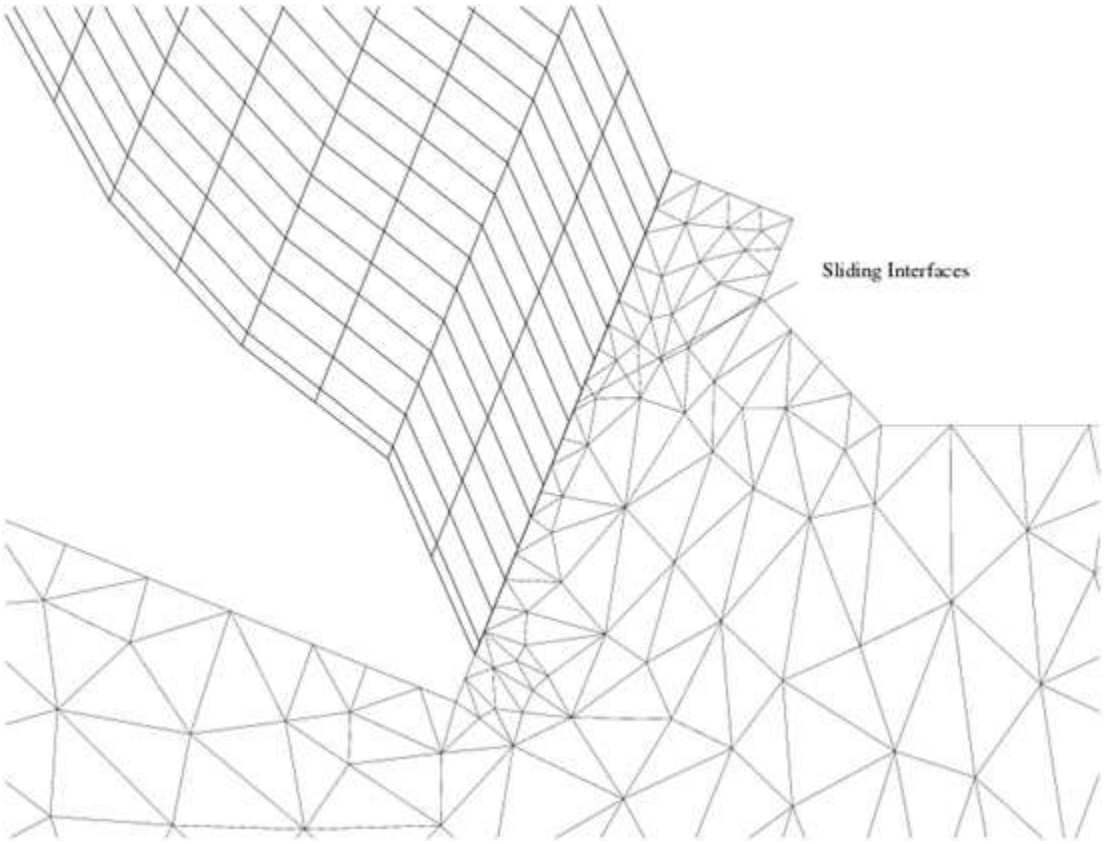
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the chamber region, you need to define a remeshing zone (triangular cells) to accommodate the various positions of the valves in the course of the simulation. In this region, the motion of the boundaries (valves and piston surfaces) is propagated to the interior nodes using the spring-based or diffusion-based smoothing method. If the cell quality violates any of the remeshing criteria that you have specified, ANSYS FLUENT will automatically agglomerate these cells and remesh them. Furthermore, ANSYS FLUENT will also remesh the deforming faces (based on the minimum and maximum length scale that you have specified) on the cylinder walls as well as those on the sliding interfaces used to connect the chamber cell zone to the layering zones above the valve surfaces.

For the intake and exhaust port regions, you can use either triangular or quadrilateral cell zones because these zones are not moving or deforming. ANSYS FLUENT will automatically mark these regions as stationary zones and will not apply any mesh motion method on these cell zones.

The dynamic layering regions above the piston and valves are conformal with the adjacent cell zone in the chamber and ports, respectively, so you do not have to use sliding interfaces to connect these cell zones together. However, you need to use sliding interfaces to connect the dynamic layering regions above the valves and the remeshing region in the chamber. This is shown in *Figure 11.55* (p. 594) with the exhaust valve almost at full extension. Notice that cells on the chamber side of the interface zone are remeshed (i.e., split or merged) as the interface zone opens and closes because of the motion of the exhaust valve.

Figure 11.55 The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone



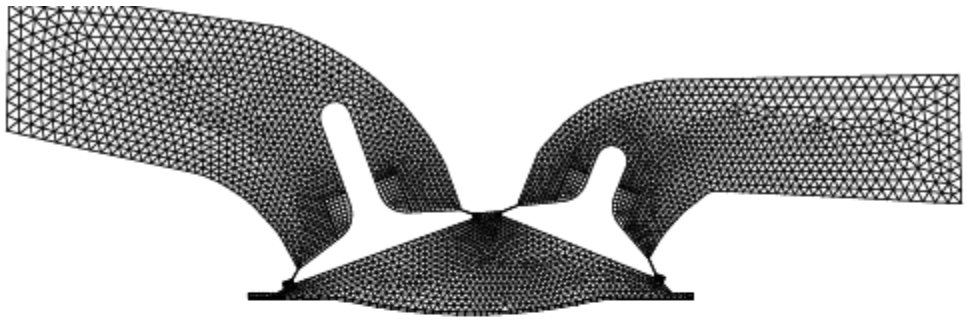
11.6.3.1.3. Defining Motion/Geometry Attributes of Mesh Zones

As the piston moves down from the TDC to the BDC position, you need to expand the remeshing region such that it can accommodate the valves when they are fully extended. To accomplish this, you need to specify the dynamic layering zone adjacent to the piston surface to move with the piston until some specified

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

distance from the TDC position. Beyond this cutoff distance, the motion of the layering zone is stopped and the piston wall is allowed to continue to the BDC position. Because there is relative motion between the piston head surface and the now non-moving dynamic layering zone, cell layers will be added when the ideal layer height criteria is violated. *Figure 11.56* (p. 595) to *Figure 11.61* (p. 598) show the sequence of meshes before and after the onset of cell layering when the motion in the layering zone above the piston surface is stopped (shown with $\Delta\theta = 5\ ^{\circ}$).

Figure 11.56 Mesh Sequence 1



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.57 Mesh Sequence 2

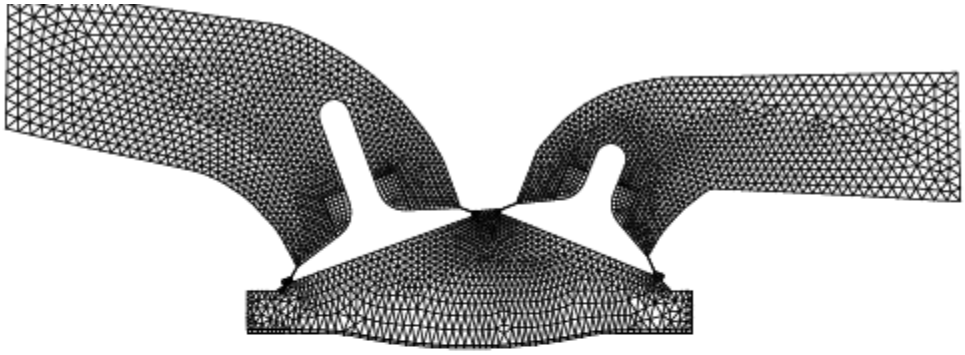
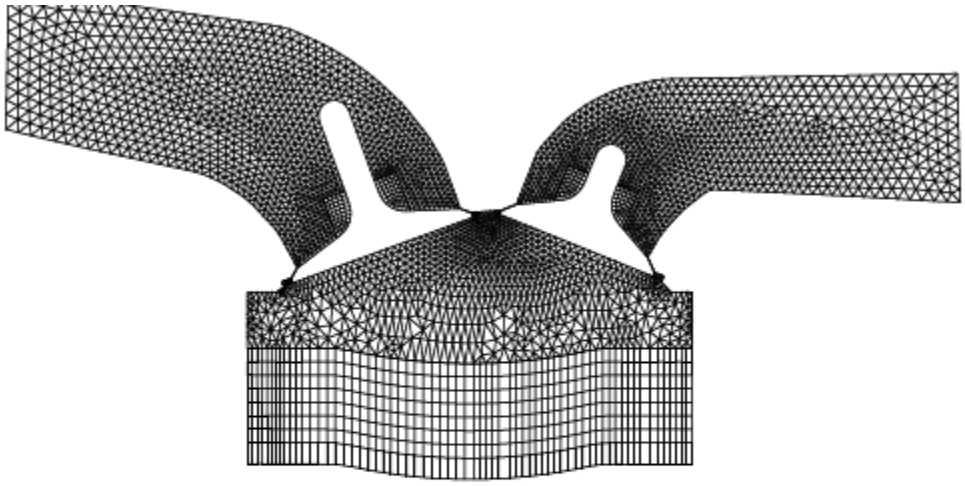


Figure 11.58 Mesh Sequence 3



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.59 Mesh Sequence 4

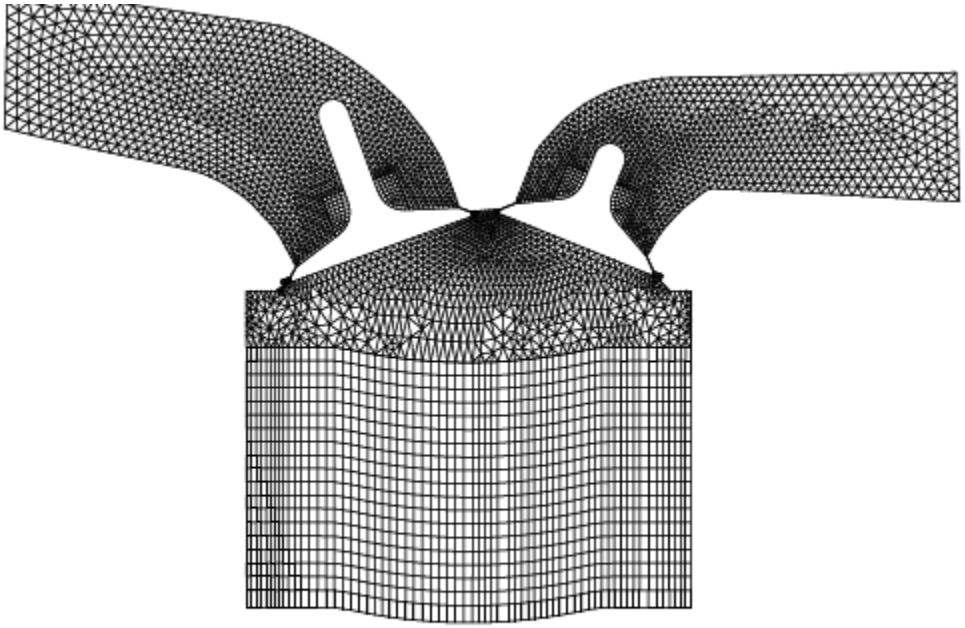
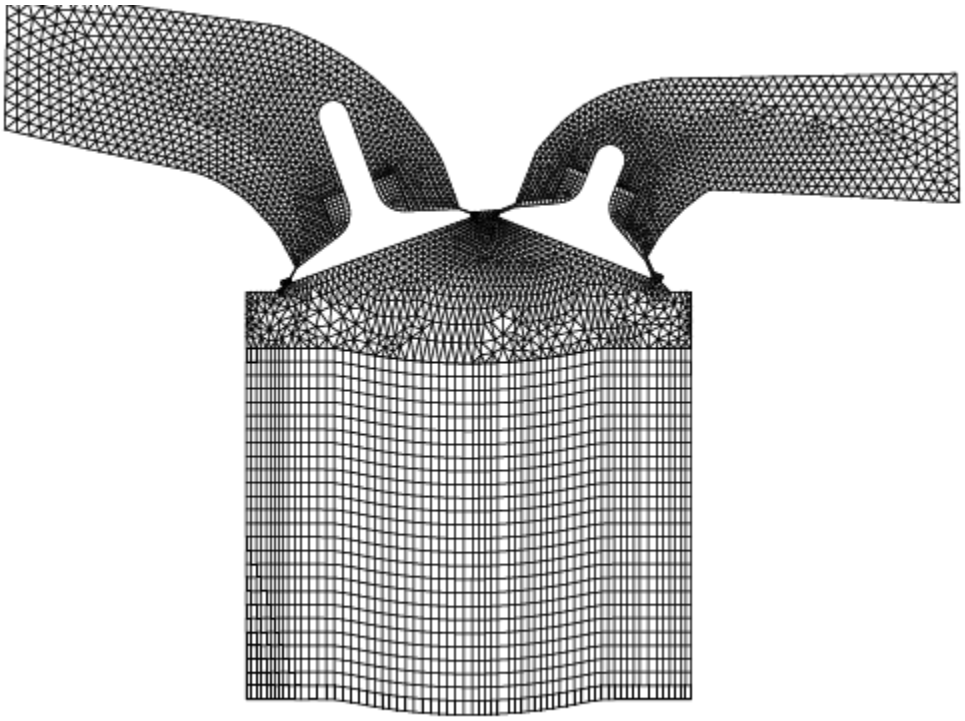
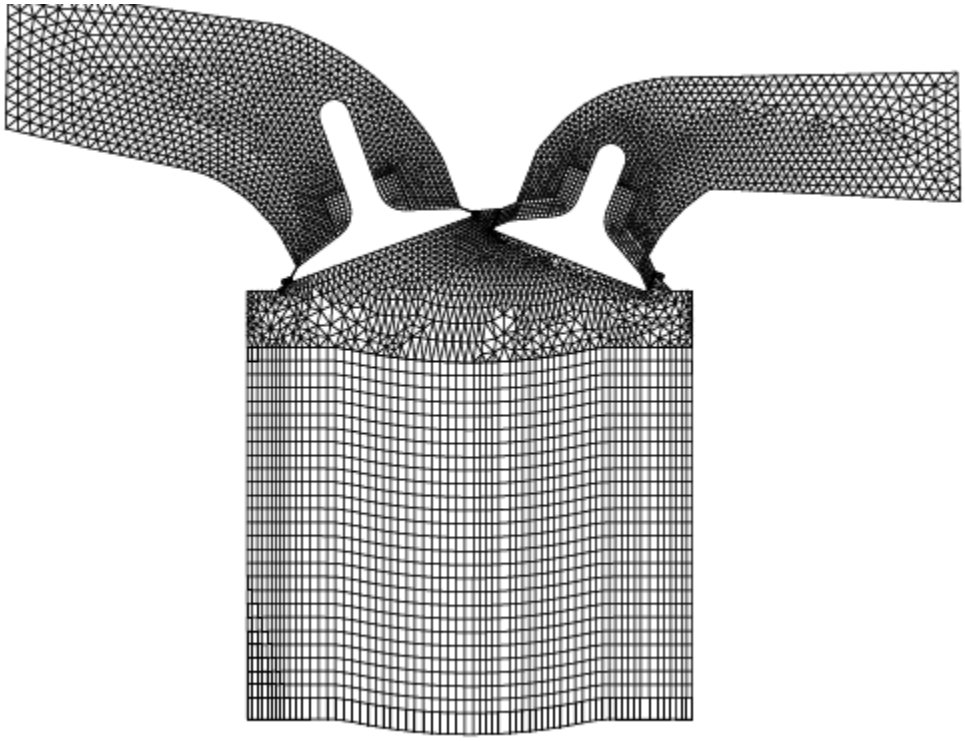


Figure 11.60 Mesh Sequence 5



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

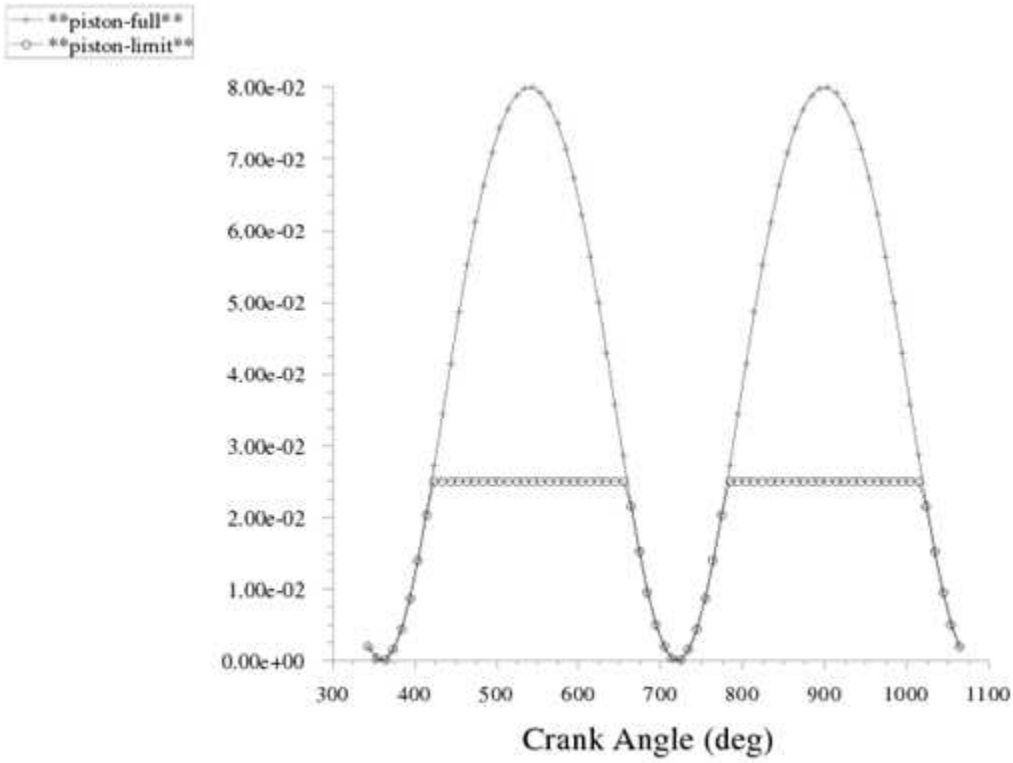
Figure 11.61 Mesh Sequence 6



ANSYS FLUENT provides built-in functions to handle the full piston motion and the limited piston motion for the dynamic layering zone above the piston surface. When you define the motion attribute of the dynamic layering zone above the piston surface, you need to use the limited piston motion function (****piston-limit**** in the **C.G. Motion UDF/Profile** field in the *Dynamic Mesh Zones Dialog Box* (p. 1889)). Note that you must define the parameters used by these functions before you can use them. In the current example, the piston stroke is 80 mm and the connecting rod length is 140 mm. The piston stroke cutoff is assumed to happen at 25 mm from TDC position. The lift as a function of crank angle between 344° and 1064° is shown in *Figure 11.62* (p. 599) for both limited and full piston motion.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.62 Piston Position (m) as a Function of Crank Angle (deg)



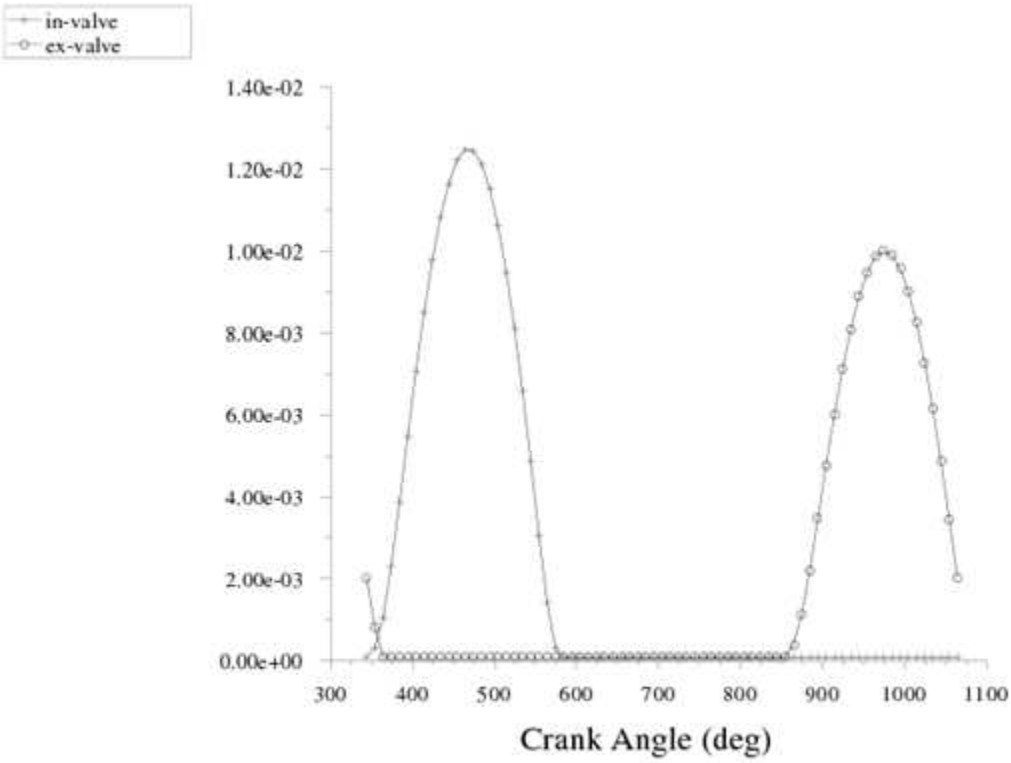
To define the motion of the valves, you need to use profiles that describe the variation of valve lift with crank angle. ANSYS FLUENT expects certain profile fields to be used to define the lift and the crank angle. For example, consider the following simplified profile definition:

```
((ex-valve 5 point)
(angle 0  180 270 360  720)
(lift 0.05 0.05 1.8 0.05 0.05))

((in-valve 5 point)
(angle 0  355  440 540 720)
(lift 0.05 0.05 2.0 0.05 0.05))
```

ANSYS FLUENT expects the `angle` and `lift` fields to define the crank angle and lift variations, respectively. The angle must be specified in degrees and the lift values must be in meters. The actual valve lift profiles that you will use for the current example are shown in *Figure 11.63* (p. 600). Notice that there is an overlapped period where both the intake and exhaust valves are open.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.63 Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg)



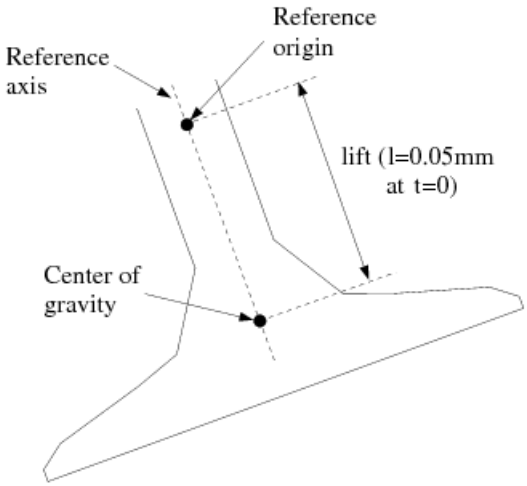
The valve lift profiles and the built-in functions will describe how each surface moves as a function of crank angle with respect to some reference point. For example, the valve lift is zero when the valve is fully closed and the valve lift is maximum when it is fully open. In order to move the surfaces, ANSYS FLUENT requires that you specify the direction of motion for each surface. ANSYS FLUENT will then update the "center of gravity" of each surface such that

$$\vec{x} = \vec{x}_{ref} - l\,\vec{e}_{axis} \qquad \textbf{(11–27)}$$

where $\vec{x}_{ref}$ is some reference position, $\vec{e}_{axis}$ is the unit vector in the direction of motion, and l is either the valve or the piston distance with respect to the reference position $\vec{x}_{ref}$. Note that the unit vector of the direction of motion is specified to point in the negative direction. For example, the correct intake valve axis for this example is $(-0.3421, 0.9397)$, as shown in *Figure 11.64* (p. 601).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.64 Definition of Valve Zone Attributes (Intake Valve)



11.6.3.1.4. Defining Valve Opening and Closure

ANSYS FLUENT assumes that once you have set up the mesh topology, the mesh topology is unchanged throughout the entire simulation. Therefore, ANSYS FLUENT does not allow you to completely close the valves such that the cells between the valve and the valve seat become degenerate (flat cells) when these surfaces come in contact (removing these flat cells would require the creation of new boundary face zones). To prevent the collapse, you need to define a minimum valve lift and ANSYS FLUENT will automatically stop the motion of the valve when the valve lift is smaller than the minimum valve lift value. The minimum valve lift value can be specified in the **In-Cylinder** tab of the *Options Dialog Box* (p. 1883). For the current example, a minimum valve lift value of 0.1 mm is assumed.

When the valve position is smaller than the minimum valve lift value, it is normal practice to assume that the valve is closed. The actual closing of the valves is accomplished by deleting the sliding interfaces that connect the chamber cell zone to the dynamic layering zones on the valves. The interface zones are then converted to walls to close off the "gaps" between the valves and the valve seats.

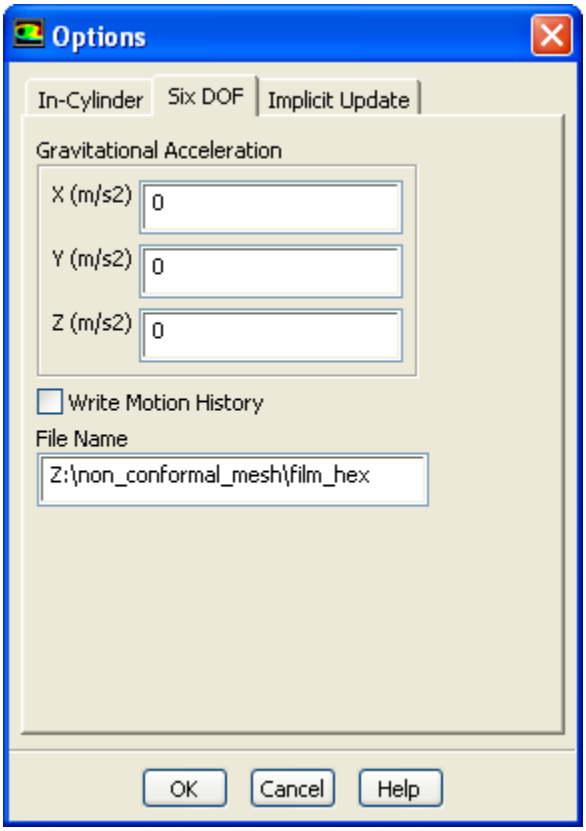
The valve opening is achieved by the reverse process. When the valve lift has reached beyond the minimum valve lift value, the valve is assumed to be open and you can redefine the sliding interfaces such that the chamber zone is now connected to the dynamic layering zones above the valves.

11.6.4. Six DOF Solver Settings

To use the six degree of freedom solver for your transient dynamic mesh simulation, select **Six DOF** under **Options** in the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.13* (p. 550)) and click the **Settings...** button. The **Options** dialog box will open, where you can click the **Six DOF** tab (*Figure 11.65* (p. 602)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.65 The Six DOF Tab of the Options Dialog Box



You can specify the gravitational acceleration in the *x*, *y*, and *z* directions either in this dialog box, or in the **Operating Conditions** dialog box. Note that you can also keep track of an object's motion history by selecting the check box next to **Write Motion History**. A single motion history file will be generated for each moving object which can be used to display zone motion for postprocessing your results. Enter the file name in the **File Name** text entry box and click **OK** (*Figure 11.65* (p. 602)).

11.6.4.1. Using the Six DOF Solver

ANSYS FLUENT's six degree of freedom (six DOF) solver computes external forces and moments such as aerodynamic and gravitational forces and moments on an object. These forces are computed by numerical integration of pressure and shear stress over the object's surfaces. Additional load forces can be added (e.g., injector forces, thrust (propulsive) forces, moments produced by a coil spring, etc.). This technique, along with the ANSYS FLUENT solver and the use of dynamic meshes, can be readily applied to many useful applications, such as store separation [*77*] (p. 2213), [*84*] (p. 2213). Note that if the mesh motion of your six DOF simulation depends on the fluid flow, it is beneficial to also enable implicit mesh updating (as described in *Implicit Update Settings* (p. 603)).

11.6.4.1.1. Setting Rigid Body Motion Attributes for the Six DOF Solver

When the **Six DOF** solver is enabled, you need to provide additional information for rigid body dynamic zones. For instance, you must use a user-defined function to define the six degrees of freedom parameters, and you must set the velocity and angular velocity for the center of gravity. For each moving object, exactly one user-defined function has to be defined, no matter how many zones there are for each object. For more information about the **Six DOF** solver settings in the *Dynamic Mesh Zones Dialog Box* (p. 1889) or rigid body motion, see *Rigid Body Motion* (p. 616).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you can also keep track of an object's motion history using the **Motion Attributes** tab.

The commands in this tab generate a single motion history file for each moving object which can be used to display zone motion for postprocessing your results. For more information on zone motion, see *Previewing the Dynamic Mesh* (p. 626).

11.6.5. Implicit Update Settings

For transient problems, you can enable implicit mesh updating when you want to have the dynamic mesh updated during a time step (as opposed to just at the beginning of a time step). This capability is beneficial only for applications in which the mesh motion depends on the flow field (e.g., cases that use the six DOF solver or involve fluid-structure interaction). For such applications, having the mesh motion updated within the time step based on the converging flow solution results in a stronger coupling between the flow solution and the mesh motion, and leads to a more robust solver run. Implicit mesh updating allows you to run simulations that otherwise could not be solved or would require an unreasonably small time step.

Note that implicit mesh updating cannot be used with the following:

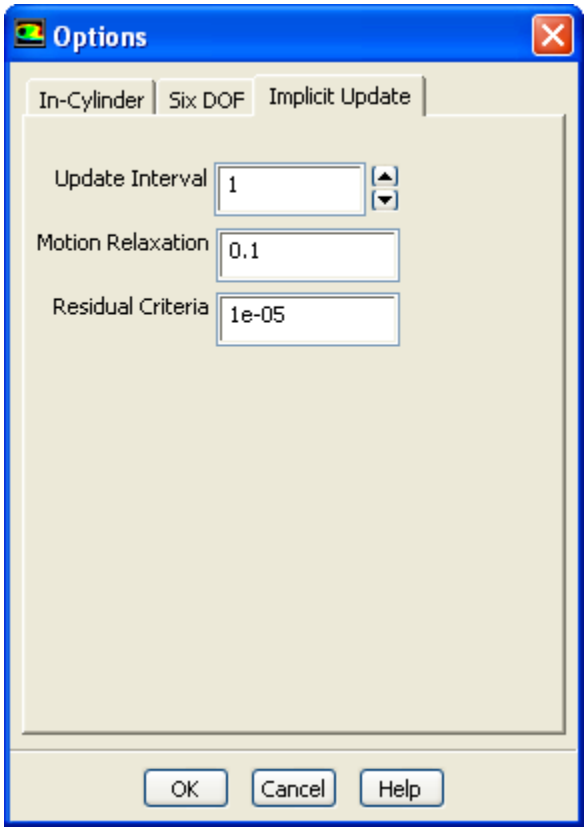
- the density-based solver when **Explicit** is selected from the **Transient Formulation** drop-down menu in the **Solution Methods** task page
- steady state solutions
- in-cylinder applications

To enable implicit mesh updating, perform the following steps:

1. Enable **Implicit Update** in the **Options** group box of the *Dynamic Mesh Task Page* (p. 1877) (*Figure 11.13* (p. 550)).
2. Click the **Settings...** button to open the *Options Dialog Box* (p. 1883).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.66 The Implicit Update Tab of the Options Dialog Box



Click the **Implicit Update** tab and enter values for the settings.

a. Enter a value for **Update Interval**, in order to specify the frequency in iterations at which the mesh will be updated within a time step.

b. Enter a value (within the range of 0 to 1) for **Motion Relaxation**, in order to define the relaxation of the motion (i.e., displacement of the nodes) during the mesh update. The relaxation of the displacements is defined by the following equation:

$$x_k = \omega(x_{computed,k}) + (1-\omega)x_{k-1} \qquad \textbf{(11–28)}$$

where x_k is the node position at iteration k (within a time step), $x_{computed,k}$ is the computed node position (based on the flow field), and ω is the motion relaxation.

c. Enter a value for **Residual Criteria**, in order to set the relative residual threshold that is used to check the motion convergence. The residual criteria is applied to a relative residual. ANSYS FLUENT scales the difference between the motion in iteration k and iteration $k-1$ by the motion computed at the beginning of the time step. If this relative motion difference is smaller than the residual criteria, the mesh motion is considered converged.

3. If you are using a UDF to compute the motion, make sure that the UDF uses the current flow field during each call to compute the motion (i.e., no previously stored information should be used). This is necessary, as the UDF will be called each time the mesh is updated — which can be several times within a time step, depending on what you entered for the **Update Interval**.

4. After you run the simulation, make sure that the motion (and consequently, the solution) is properly converged. If the motion requires more iterations to converge than the flow field, a warning will be

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

displayed in the console during the iteration process. Note that the maximum number of iterations per time step (defined in the **Run Calculation** task page) is respected by the mesh motion convergence check.

11.6.6. Defining Dynamic Mesh Events

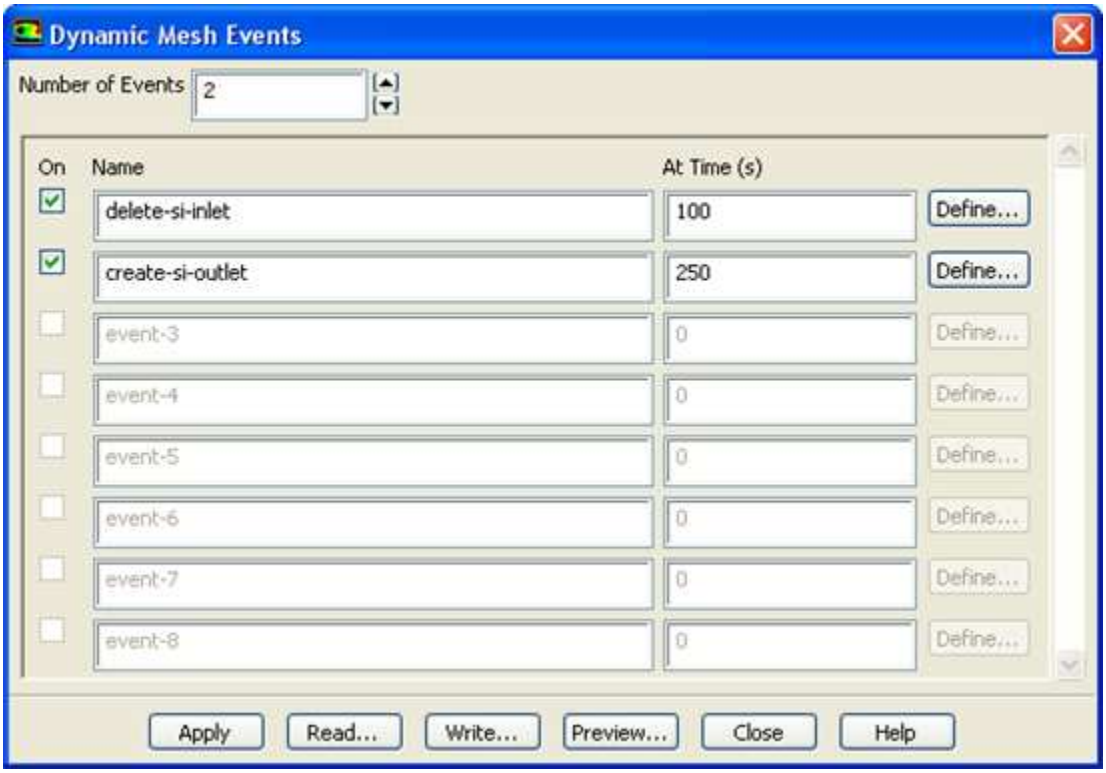
If you are simulating a transient flow, you can use the events in ANSYS FLUENT to control the timing of specific events during the course of the simulation. With in-cylinder flows for example, you may want to open the exhaust valve (represented by a pair of deforming sliding interfaces) by creating an event to create the sliding interfaces at some crank angle. You can also use dynamic mesh events to control when to suspend the motion of a face or cell zone by creating the appropriate events based on the crank angle or time. Note that in-cylinder flows are crank angle-based, whereas all other flows are time-based.

11.6.6.1. Procedure for Defining Events

You can define the events using the *Dynamic Mesh Events Dialog Box* (p. 1886) (*Figure 11.67* (p. 605)).

Dynamic Mesh → **Events...**

Figure 11.67 The Dynamic Mesh Events Dialog Box



The procedure for defining events is as follows:

1. Increase the **Number of Events** value to the number of events you wish to specify. As this value is increased, additional event entries in the dialog box will become editable.
2. Enable the check box next to the first event and enter a name for the event under the **Name** heading.
3. Specify either the time or the crank angle at which you want the event to occur.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

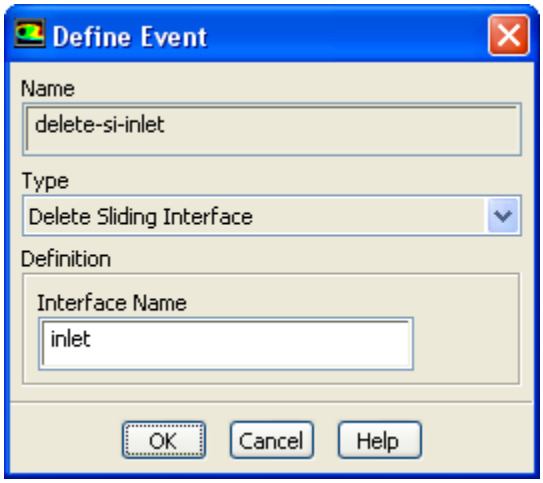
For in-cylinder flows, specify the crank angle at which you want the event to occur under **At Crank Angle**.

For non-in-cylinder flows, specify the time (in seconds) at which you want the event to occur under **At Time**.

It is not necessary to specify the events in order of increasing time or crank angle, but it may be easier to keep track of events if you specify them in the order of increasing time or angle.

4. Click the **Define...** button to open the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)).

Figure 11.68 The Define Event Dialog Box



5. In the *Define Event Dialog Box* (p. 1887), choose the type of event by selecting **Change Zone Type**, **Copy Zone BC**, **Activate Cell Zone**, **Deactivate Cell Zone**, **Create Sliding Interface**, **Delete Sliding Interface**, **Change Motion Attribute**, **Change Time Step Size**, **Change Under-Relaxation Factors**, **Insert Boundary Zone Layer**, **Remove Boundary Zone Layer**, **Insert Interior Zone Layer**, **Remove Interior Zone Layer**, **Insert Cell Layer**, **Remove Cell Layer**, **Execute Command**, or **Replace Mesh** in the **Type** drop-down list. These event types and their definitions are described later in this section.
6. Repeat steps 2–5 for the other events, if relevant.
7. Click **Apply** in the *Dynamic Mesh Events Dialog Box* (p. 1886) after you finish defining all events.
8. To play the events to check that they are defined correctly, click the **Preview...** button in the *Dynamic Mesh Events Dialog Box* (p. 1886). This displays the *Events Preview Dialog Box* (p. 1889).

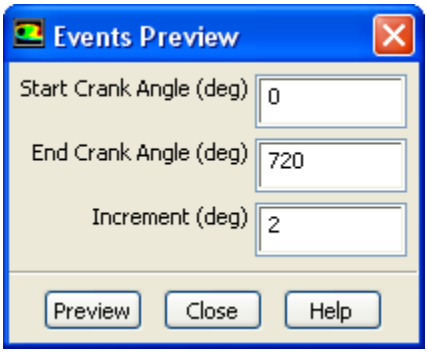
   For in-cylinder flows, you use the *Events Preview Dialog Box* (p. 1889) (*Figure 11.69* (p. 607)), to enter the crank angles at which you want to start and end the playback in the **Start Crank Angle** and **End Crank Angle** fields, respectively.

   For non-in-cylinder flows, you use the *Events Preview Dialog Box* (p. 1889) to enter the time at which you want to start and end the playback in the **Start Time** and **End Time** fields, respectively.

   Specify the size of the step to take during the playback in the **Increment** field. Click **Preview** to play back the events. ANSYS FLUENT will play the events at the time (or crank angle in the case of in-cylinder flows) specified for each event and report when each event occurs in the text (console) window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.69 The Events Preview Dialog Box for In-Cylinder Flows



For in-cylinder simulations, you need to specify the events for one complete engine cycle. In the subsequent cycles, the events are executed whenever

$$\theta_{event}=\theta_c \pm n\theta_{period} \tag{11–29}$$

where θ_{event} is the event crank angle, θ_c is the current crank angle calculated from *Equation 11–25* (p. 588), θ_{period} is the crank angle period for one cycle, and $n$is some integer.

As an example, for in-cylinder simulations, you are not required to specify the event crank angle to correspond exactly to the current crank angle calculated from *Equation 11–25* (p. 588). ANSYS FLUENT will execute an event if the current crank angle is between $\pm 0.5\Delta\theta$ where $\Delta\theta$ is the equivalent change in crank angle for the time step. For example, if the event preview is executed between crank angle of 340° and 1060° (crank period is 720°) using an increment of 1°, ANSYS FLUENT will report the following in the text window.

```
Execute Event: open-in-valve-left (defined at: 353.10, current angle: 353.00)
Execute Event: open-in-valve-right (defined at: 353.00, current angle: 353.00)
Execute Event: close-ex-valve-right (defined at: 355.60, current angle: 356.00)
Execute Event: close-ex-valve-left (defined at: 357.80, current angle: 358.00)
Execute Event: close-in-valve-left (defined at: 571.60, current angle: 572.00)
Execute Event: close-in-valve-right (defined at: 571.80, current angle: 572.00)
Execute Event: open-ex-valve-right (defined at: 137.10, current angle: 857.00)
Execute Event: open-ex-valve-left (defined at: 139.00, current angle: 859.00)
```

Notice that events defined at 137.10° and 139° are executed at 857° and 859°, respectively, because they satisfy the condition of *Equation 11–29* (p. 607).

11.6.6.2. Defining Events for In-Cylinder Applications

ANSYS FLUENT will automatically limit the valve lift values depending on the specified minimum valve lift value. However, the conversion of the sliding interface zones to walls (and vice versa) is accomplished via the in-cylinder events (see *Defining Dynamic Mesh Events* (p. 605)). For example, if the exhaust valve closes at -5° before TDC position, you must define a **Delete Sliding Interface** event at the crank angle of -5°. You need to define similar events for the intake valve opening (using the **Create Sliding Interface** event), the intake valve closing (**Delete Sliding Interface** event), and the exhaust valve opening (**Create Sliding Interface** event) at the respective crank angles.

For the current example, the exhaust valve is assumed to be open between 131° and 371° and the intake valve is open between at 345° and 584°.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.6.2.1. Events

Each of the available events is described below.

11.6.6.2.2. Changing the Zone Type

You can change the type of a zone to be a wall, or an interface, interior, fluid, or solid zone during your simulation. To change the type of a zone, select **Change Zone Type** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)). Select the zone(s) that you want to change in the **Zone** list, and then select the new zone type in the **New Zone Type** drop-down list.

11.6.6.2.3. Copying Zone Boundary Conditions

You can copy boundary conditions from one zone to other zones during your simulation. If, for example, you have changed an inlet zone to type wall with the **Change Zone Type** event, you can set the boundary conditions of the new zone type by simply copying the boundary conditions from a known zone with the corresponding zone type.

To copy boundary conditions from one zone to another, select **Copy Zone BC** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)). In the **From Zone** drop-down list, select the zone that has the conditions you want to copy. In the **To Zone(s)** list, select the zone or zones to which you want to copy the conditions.

ANSYS FLUENT will set *all* of the boundary conditions for the zones selected in the **To Zone(s)** list to be the same as the conditions for the zone selected in the **From Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Note that you cannot copy conditions from external walls to internal (i.e., two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

11.6.6.2.4. Activating a Cell Zone

To activate a cell zone, select **Activate Cell Zone** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)), then select the zone that you want to activate in the **Zone(s)** list. For more information, see *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195).

11.6.6.2.5. Deactivating a Cell Zone

To deactivate a cell zone, select **Deactivate Cell Zone** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)), then select the zone that you want to deactivate in the **Zone(s)** list.

Only deactivated zones can be activated. When a zone is deactivated, ANSYS FLUENT skips the zone during the calculations. For more information, see *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195).

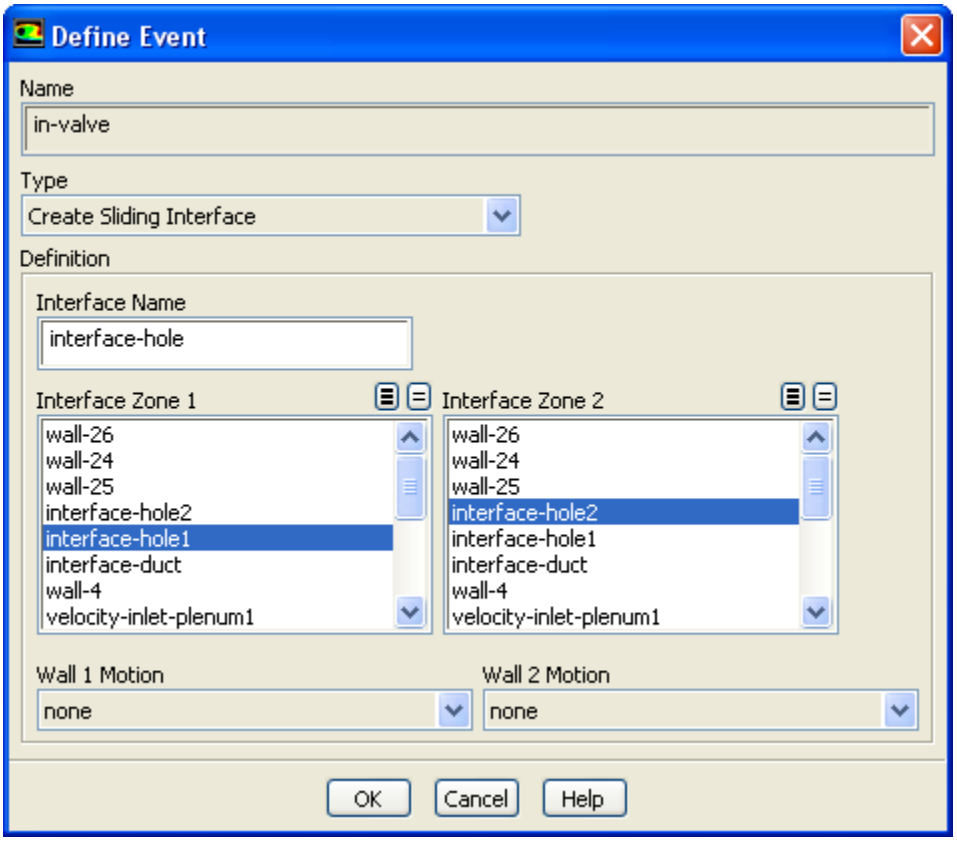
11.6.6.2.6. Creating a Sliding Interface

To create a sliding interface during your simulation, select **Create Sliding Interface** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.70* (p. 609)). Enter a name for the sliding interface in the **Interface Name** field. Select the zones on either side of the interface in the **Interface Zone 1** and **Interface Zone 2** drop-down lists.

You have the option to select any number of zones listed under each of the interface zones. ANSYS FLUENT calculates intersections between all possible combinations of the left and right side of the interfaces, allowing you more flexibility in terms of creating zones and defining the interfaces.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.70 The Define Event Dialog Box for the Creating Sliding Interface Option



Important

If ANSYS FLUENT finds another interface with the same name as defined in the event, then the old interface will be deleted and a new one created as defined in the dynamic mesh event.

If the interface zones that you selected above do not overlap each other completely, the non-overlapped regions on each interface zones are put into separate wall zones by ANSYS FLUENT. If these wall zones (i.e., non-overlapped regions) have motion attributes associated with them, their motion can only be specified by copying the motion from another dynamic zone by selecting the appropriate dynamic zones in the **Wall 1 Motion** and **Wall 2 Motion** drop-down lists, respectively.

Note that you don't have to change the boundary type from wall to interface. When the **Create Sliding Interface** event is executed, ANSYS FLUENT will automatically change the boundary type of the face zones selected in **Interface Zone 1** and **Interface Zone 2** to type interface before the sliding interface is created.

11.6.6.2.7. Deleting a Sliding Interface

To delete a sliding interface that has been created earlier in your in-cylinder simulation, select **Delete Sliding Interface** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)). Enter the name of the sliding interface to be deleted in the **Interface Name** field.

As with the **Create Sliding Interface** event, ANSYS FLUENT will automatically change the corresponding interface zones to wall. However, you may want to use the **Copy Zone BC** event to set any boundary conditions that are not the default conditions that ANSYS FLUENT assumes.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.6.2.8. Changing the Motion Attribute of a Dynamic Zone

To change the motion attribute of a dynamic zone during your in-cylinder calculation, select **Change Motion Attribute** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)). Select the **Attribute** (**slide**, **moving**, or **remesh**) and set the appropriate **Status** (**enable** or **disable**). Select the corresponding dynamic zones for which you want to change the motion attributes in the **Dynamic Zones** list.

The **slide** attribute is used to enable or disable smoothing of nodes on selected deforming face zones, the **moving** attribute is used to suspend the motion of selected moving zones, and the **remesh** attribute is used to enable and disable face remeshing on selected deforming face zones.

11.6.6.2.9. Changing the Time Step

To change the time step at some point during the simulation, select **Change Time Step Size** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the new physical time step size by entering the new **Time Step Size** in seconds.

For in-cylinder simulations, specify the new physical time step by entering the new **Crank Angle Step Size** value in degrees. The physical time step is calculated from

$$\Delta t = \frac{\Delta\theta_c}{6\Omega_{shaft}} \tag{11–30}$$

where the unit of Ω_{shaft} is assumed to be in RPM.

11.6.6.2.10. Changing the Under-Relaxation Factor

To change one or more under-relaxation factors, select **Change Under-Relaxation Factor** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)). Select the under-relaxation factor that you wish to change, and assign a new value to it in the **Under-Relaxation Factors** list. For more information on setting under-relaxation factors, see *Setting Under-Relaxation Factors* (p. 1217).

11.6.6.2.11. Inserting a Boundary Zone Layer

To insert a new cell zone layer as a separate cell zone adjacent to a boundary, select **Insert Boundary Zone Layer** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the **Base Dynamic Zone**, from which the layer of cells is to be created, and the **Side Dynamic Zone**, which represents the deforming face zone adjacent to the **Base Dynamic Zone** before the layer is inserted. The new cell zone will inherit the boundary conditions of the cell zone adjacent to the **Base Dynamic Zone** before the layer is inserted.

Note that a new cell layer can be inserted only from a one-sided **Base Dynamic Zone**. You cannot insert a new cell layer from an interior face zone.

Figure 11.71 (p. 611) and *Figure 11.72* (p. 611) illustrate the insertion of a boundary zone layer. In both figures, the circular face at the top of the cylinder is the base dynamic zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.71 Boundary Zone Before Insertion

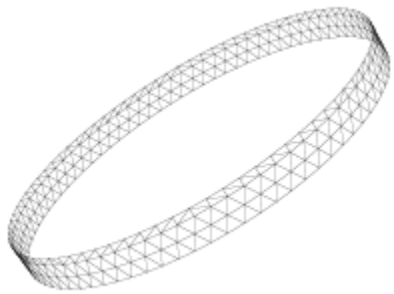


Figure 11.72 Boundary Zone After Insertion

11.6.6.2.12. Removing a Boundary Zone Layer

To remove the cell zone layer inserted using the **Insert Boundary Zone Layer** event, select **Remove Boundary Zone Layer** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the same **Base Dynamic Zone** that you used when you defined the insert boundary layer event.

Note that a cell layer can be removed only from a one-sided **Base Dynamic Zone**.

11.6.6.2.13. Inserting an Interior Zone Layer

To insert a new zone layer as a separate cell zone adjacent to the internal side of a boundary, select **Insert Interior Zone Layer** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the **Base Dynamic Zone** and the **Side Dynamic Zone** as described in the **Insert Boundary Zone Layer** event. You also need to specify the names of the new interior face zones (**Internal Zone 1 Name** and **Internal Zone 2 Name**) that will be created after the cell zone layer is created by ANSYS FLUENT.

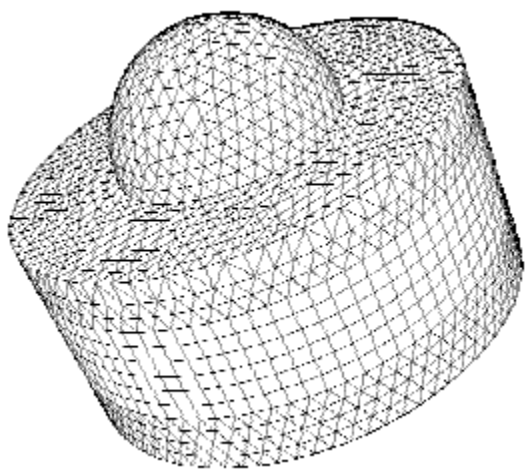
ANSYS FLUENT inserts the interior cell layer by splitting the cell zone adjacent to the **Base Dynamic Zone** with a plane. The position of the plane and the normal direction of the plane are implicitly defined by the cylinder origin and cylinder axis of the **Side Dynamic Zone**.

Figure 11.73 (p. 612) and *Figure 11.74* (p. 612) illustrate the insertion of an interior zone layer.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.73 Interior Zone Before Insertion

Figure 11.74 Interior Zone After Insertion



11.6.6.2.14. Removing an Interior Zone Layer

To remove the zone layer inserted using the **Insert Interior Zone Layer** event, select **Remove Interior Zone Layer** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the same **Internal Zone 1 Name** and **Internal Zone 2 Name** that you used to define the **Insert Interior Zone Layer** event.

11.6.6.2.15. Inserting a Cell Layer

To manually insert a new cell layer to the existing cell zone, select **Insert Cell Layer** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the **Adjacent Dynamic Face Zone** and the **Direction Parameter**. This can only work on zones that are suited for layering (see *Applicability of the Dynamic Layering Method* (p. 567)).

11.6.6.2.16. Removing a Cell Layer

To manually remove a cell layer from an existing cell zone, select **Remove Cell Layer** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887). Specify the **Adjacent Dynamic Face Zone** and the **Direction**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter. This can only work on zones that are suited for layering (see *Applicability of the Dynamic Layering Method* (p. 567)).

11.6.6.2.17. Executing a Command

To execute a command, select **Execute Command** in the **Type** drop-down list in the *Define Event Dialog Box* (p. 1887) (*Figure 11.68* (p. 606)). A command can be a series of text or Scheme commands, or a macro you have defined (or will define) using the *Define Macro Dialog Box* (p. 1951) (see *Defining Macros* (p. 1290)). Enter the series of commands or the name of the macro in the **Command** text-entry box.

Important

If the command to be executed involves saving a file, see *Saving Files During the Calculation* (p. 1291) for important information.

11.6.6.2.18. Replacing the Mesh

To replace the mesh and interpolate existing data onto the new mesh during your simulation, select **Replace Mesh** from the **Type** drop-down list in the **Define Event** dialog box (*Figure 11.68* (p. 606)). Then, specify the replacement mesh under **Mesh File**. Enable **Interpolate Data Across Zones** if necessary (see *Replacing the Mesh* (p. 198) for details).

11.6.6.3. Exporting and Importing Events

If you want to save the events you have defined to a file, click **Write...** in the *Dynamic Mesh Events Dialog Box* (p. 1886) and specify the **Event File** in *The **Select File** Dialog Box* (p. 33).

To read the events back into ANSYS FLUENT, click **Read...** in the *Dynamic Mesh Events Dialog Box* (p. 1886) and specify the **Event File** in *The **Select File** Dialog Box* (p. 33).

11.6.7. Specifying the Motion of Dynamic Zones

You need to define the motion of the dynamic zones in your model. If the zone is a rigid body, you can use a profile or user-defined function (UDF) to define the motion of the rigid body or use the six DOF solver. If the zone is a deforming zone, you can define the geometry and the parameters that control the face or zone remeshing, if applicable. For a zone that is deforming and moving at the same time, you can use a user-defined function to define the geometry and motion of the zone as they change with time.

11.6.7.1. General Procedure

You will specify the motion of the dynamic zones in your model using the **Dynamic Mesh Zones** dialog box

Dynamic Mesh → **Create/Edit...**

Details about specifying different types of motion are provided in this section.

11.6.7.1.1. Creating a Dynamic Zone

When you have completed the specification of a dynamic zone, click **Create** in the *Dynamic Mesh Zones Dialog Box* (p. 1889) to complete the specification and add the zone to the **Dynamic Mesh Zones** list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.7.1.2. Modifying a Dynamic Zone

If you want to make a change to the specification of a dynamic zone, select the zone in the **Dynamic Mesh Zones** list, change the specification, and then click **Create** in the *Dynamic Mesh Zones Dialog Box* (p. 1889) to update the specification.

11.6.7.1.3. Checking the Center of Gravity

If a dynamic zone has solid body motion, you can view its current position and orientation of the center of gravity (with respect to initial data) by selecting the zone in the **Dynamic Mesh Zones** list and viewing the values under **Center of Gravity Location** and **Center of Gravity Orientation**.

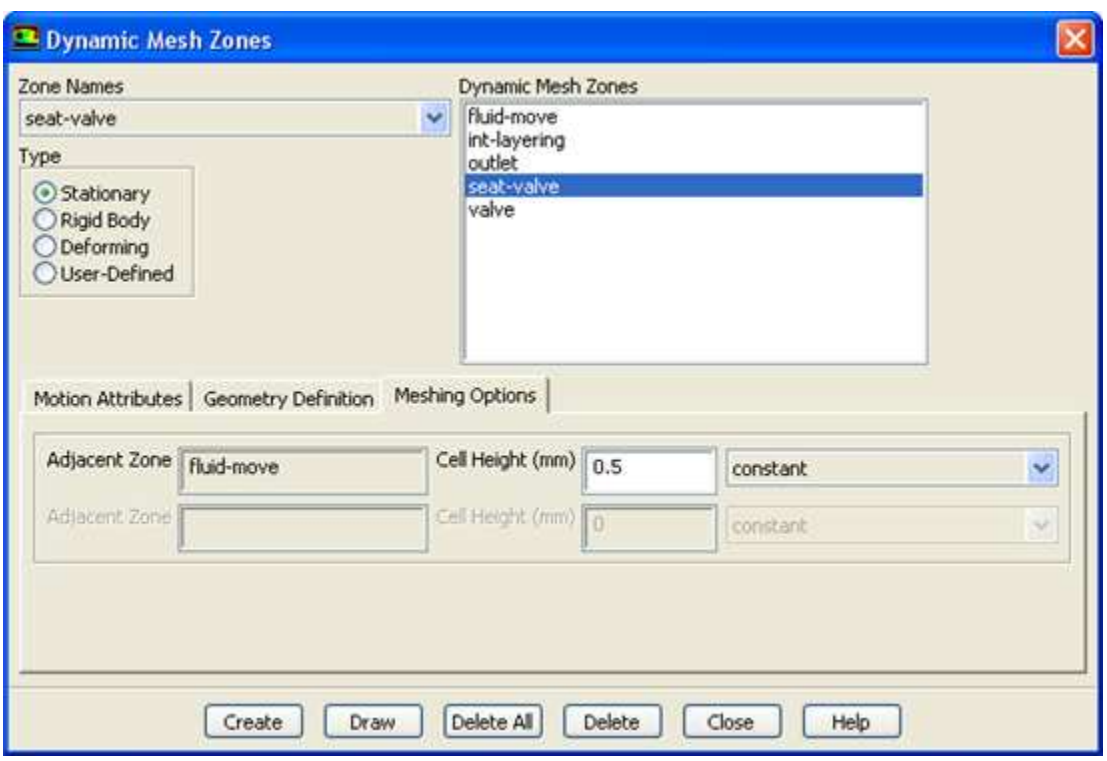
11.6.7.1.4. Deleting a Dynamic Zone

To delete a dynamic zone that you have specified, select the zone in the **Dynamic Mesh Zones** list, and click **Delete** or **Delete All**. The zone or zones will be removed from the **Dynamic Mesh Zones** list.

11.6.7.2. Stationary Zones

By default, if no motion (moving or deforming) attributes are assigned to a face or cell zone, then the zone is not considered when updating the mesh to the next time step. However, there are cases where an explicit declaration of a stationary zone is required. For example, if a cell zone is assigned some solid body motion, the positions of all nodes belonging to the cell zone will be updated even though some of the nodes may also be part of a non-moving boundary zone. An explicit declaration of a stationary zone excludes the nodes on these zones when updating the node positions.

Figure 11.75 The Dynamic Mesh Zones Dialog Box for a Stationary Zone



To define a stationary zone in your model, follow the steps below.

1. Select the stationary zone in the **Zone Names** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select **Stationary** under **Type**.
3. If the stationary zone is a face zone that is not a boundary of a CutCell dynamic cell zone, then define the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in *Equation 11–14* (p. 565) and *Equation 11–15* (p. 566) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS FLUENT, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the UDF Manual for information about UDFs.
4. If the stationary zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab. See *Figure 11.76* (p. 616) for an example of the **Dynamic Mesh Zones** dialog box with the **Meshing Options** tab as it appears for CutCell boundary zones. (The same meshing input is required at all CutCell boundary zones, whether the motion type is stationary, rigid body, deforming, or user-defined.)

   Two different size functions are available to control the local mesh refinement:

   - soft size function

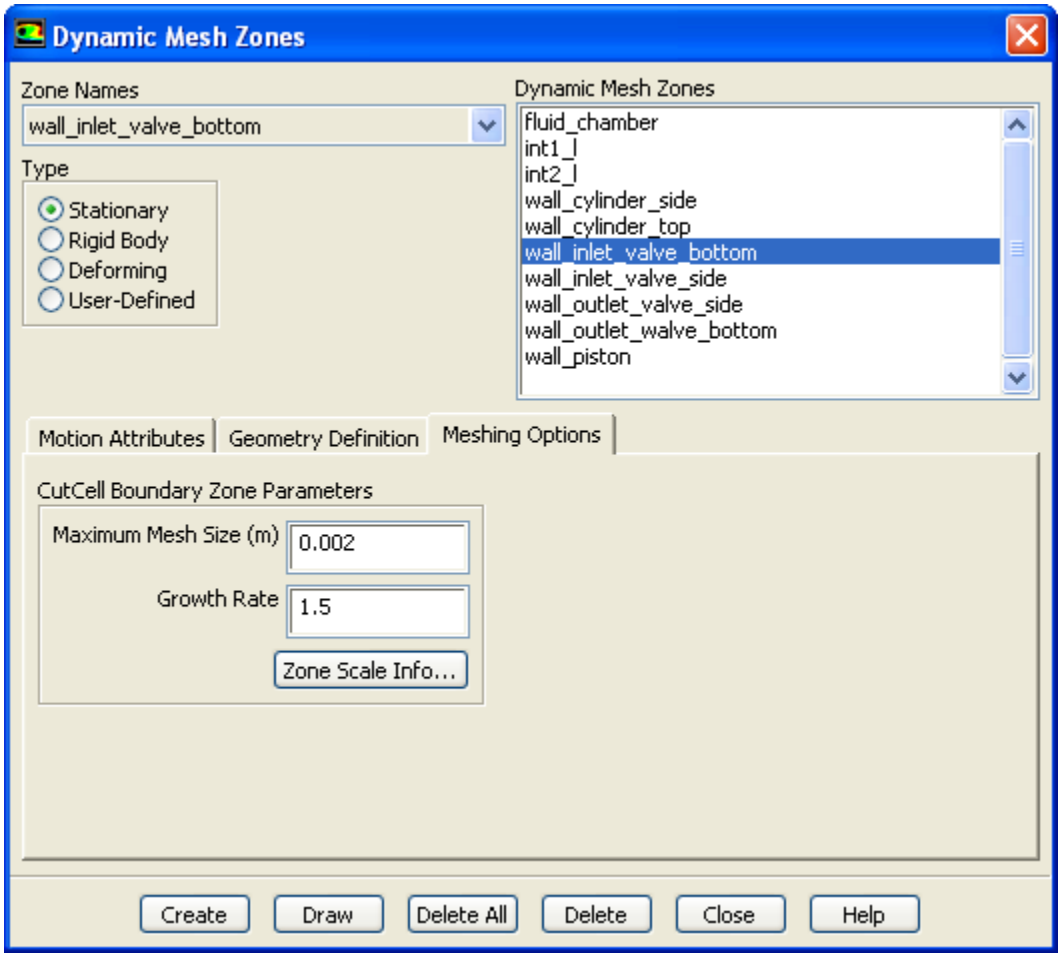
     For this size function, the Cartesian mesh is locally refined such that the resulting mesh size is smaller or equal to the **Maximum Mesh Size** specified.
   - mesh-based size function

     For this size function, the Cartesian mesh is locally refined such that the resulting mesh size is locally of the same size as the input mesh provided.

   The switch between using the soft or the mesh-based size function depends on the value entered for the **Maximum Mesh Size**. A value of `0` specifies that the mesh-based size function is used. If a positive value is entered, then the soft size-function is used and the value entered locally limits the mesh size. The **Growth Rate** controls the rate at which the Cartesian mesh grows away from the boundary. The maximum Cartesian mesh size obtained in the cell zone is always controlled by the specified **Maximum Mesh Size** for the CutCell cell zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.76 The Dynamic Mesh Zones Dialog Box for a CutCell Boundary Zone



You can view the vital statistics of your zone by clicking on the **Zone Scale Info...** button. This opens the *Zone Scale Info Dialog Box* (p. 1894), where you can view the minimum, maximum, and average length scale values, as well as the maximum skewness value.

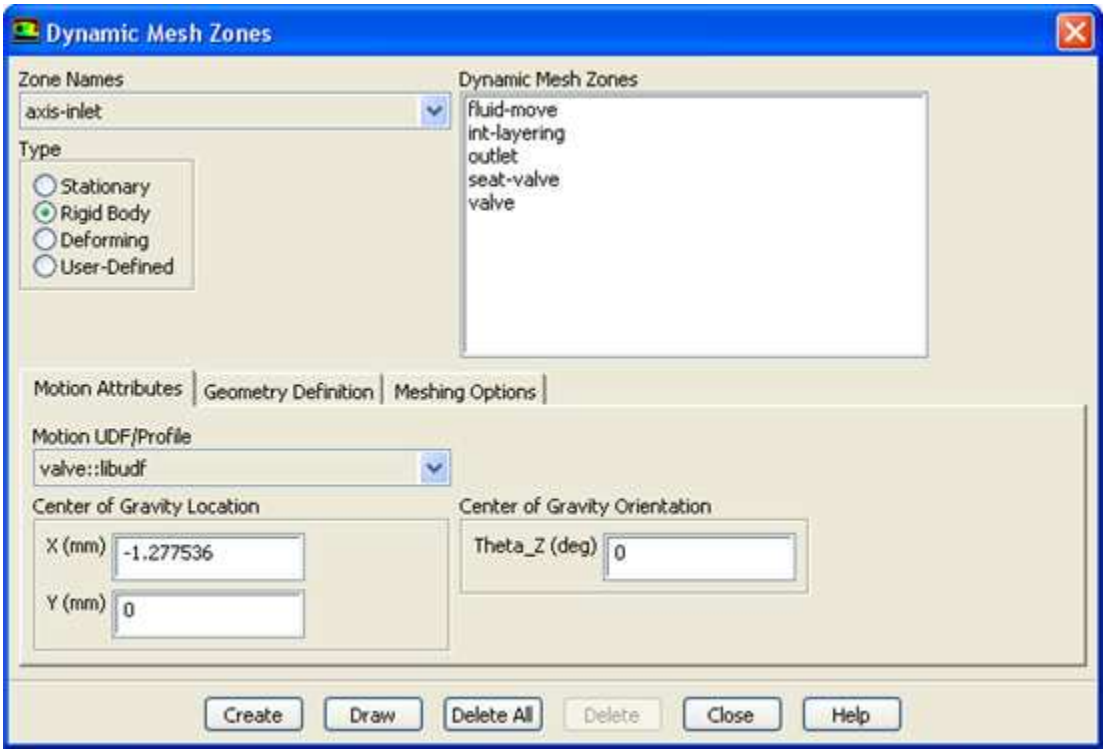
5. Click **Create**.

11.6.7.3. Rigid Body Motion

To define a rigid-body zone in your model, follow the steps below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.77 The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion



1. Select the rigid body zone in the **Zone Names** drop-down list.
2. Select the **Rigid Body** option under **Type**.
3. If you want to specify the motion of the rigid body zone using a profile or user-defined function, then select a profile or user-defined function from the **Motion UDF/Profile** drop-down list in the **Motion Attributes** tab. See *Profiles* (p. 366) and *Solid-Body Kinematics* (p. 623) for information on profiles, and see the UDF Manual for information on user-defined functions.
4. If you want to use the **Six DOF** solver option, then select the appropriate UDF from the **Six DOF UDF** drop-down list in the **Motion Attributes** tab (see *Figure 11.78* (p. 618)). Note that you should make sure that **On** is enabled under **Six DOF Options** to ensure that the Six DOF solver is being used. See the UDF Manual for information on user-defined functions. For more information about the six DOF solver, see *Using the Six DOF Solver* (p. 602).

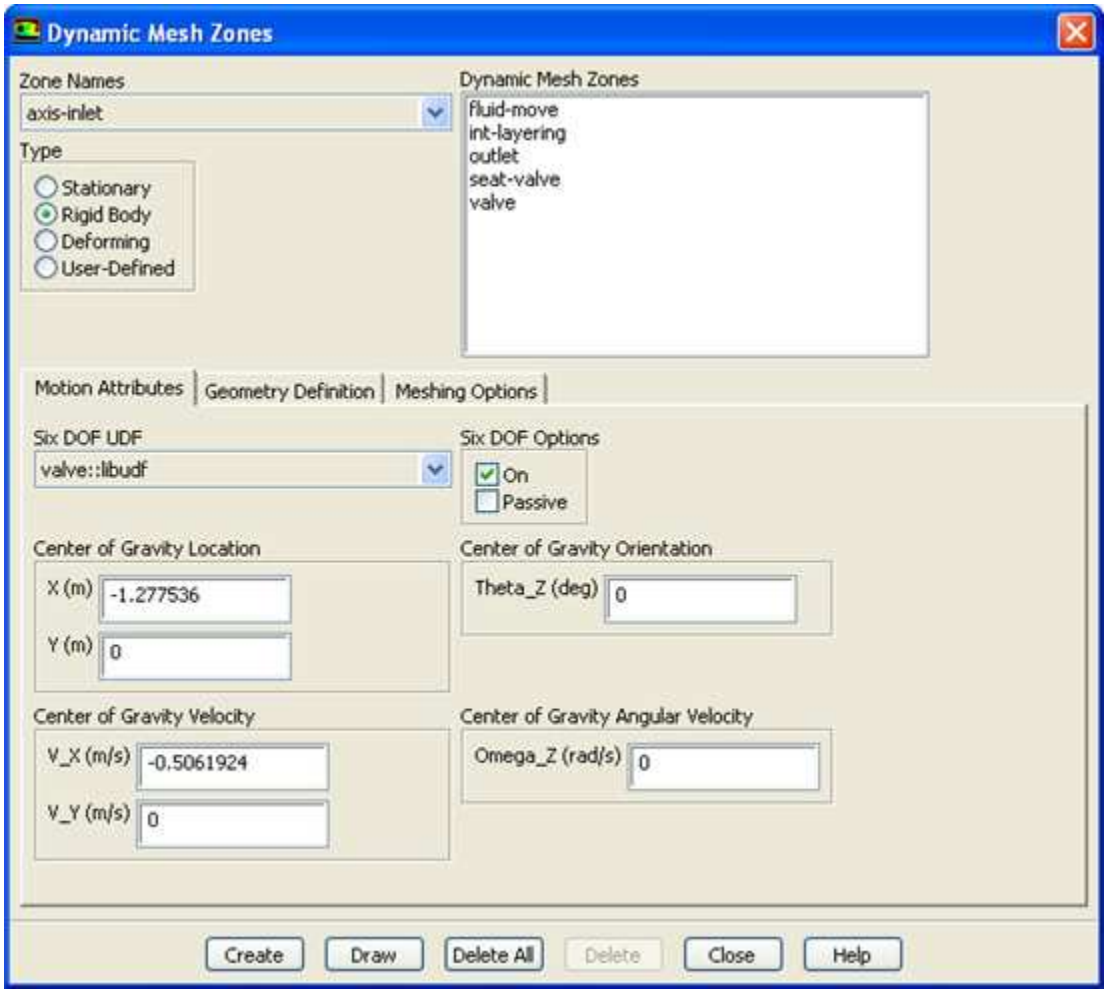
   Note that the **Passive** option under **Six DOF Options** is used when you do not want the forces and moments on the zone to be taken into consideration.
5. Specify the initial location of the center of gravity for the rigid body by entering the coordinates of the center of gravity in **Center of Gravity Location**.
6. Specify the orientation of the object with respect to the center of gravity (in the inertia coordinate system) by entering the orientations of the center of gravity in **Center of Gravity Orientation**.

   For most cases, this is an initial reference orientation that ANSYS FLUENT later updates, letting you keep track of the object's current orientation. The center of gravity orientation is most useful when using the Six DOF solver, where it is used to compute the transformation matrices (Six DOF (6DOF) Solver Theory in the Theory Guide).
7. When using the Six DOF solver, specify the velocity of the center of gravity with respect to the inertia coordinate system by entering the velocity of the center of gravity in **Center of Gravity Velocity**. Also,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specify the angular velocity of the center of gravity with respect to the inertia coordinate system by entering the angular velocity of the center of gravity in **Center of Gravity Angular Velocity**.

Figure 11.78 The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver



8. If you are solving an in-cylinder problem, specify the direction of the reference axis of the valves or piston in **Valve/Piston Axis**.

   The current valve lift or piston stroke is automatically updated in **Lift/Stroke** when you click **Create** based on the parameters you have specified earlier when you first invoke the in-cylinder option.

9. If the rigid body zone is a face zone that is not a boundary of a CutCell dynamic cell zone, specify the **Cell Height** for each **Adjacent Zone** in the **Meshing Options** tab. The **Cell Height** is the ideal cell height (h_{ideal} in *Equation 11–14* (p. 565) and *Equation 11–15* (p. 566) that is used by ANSYS FLUENT to determine when the prismatic layer next to the rigid body should be split or merged with the layer next to it. If the adjacent zone is tetrahedral or triangular, the ideal height is used by ANSYS FLUENT to determine if adjacent cells need to be agglomerated for local remeshing. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS FLUENT, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

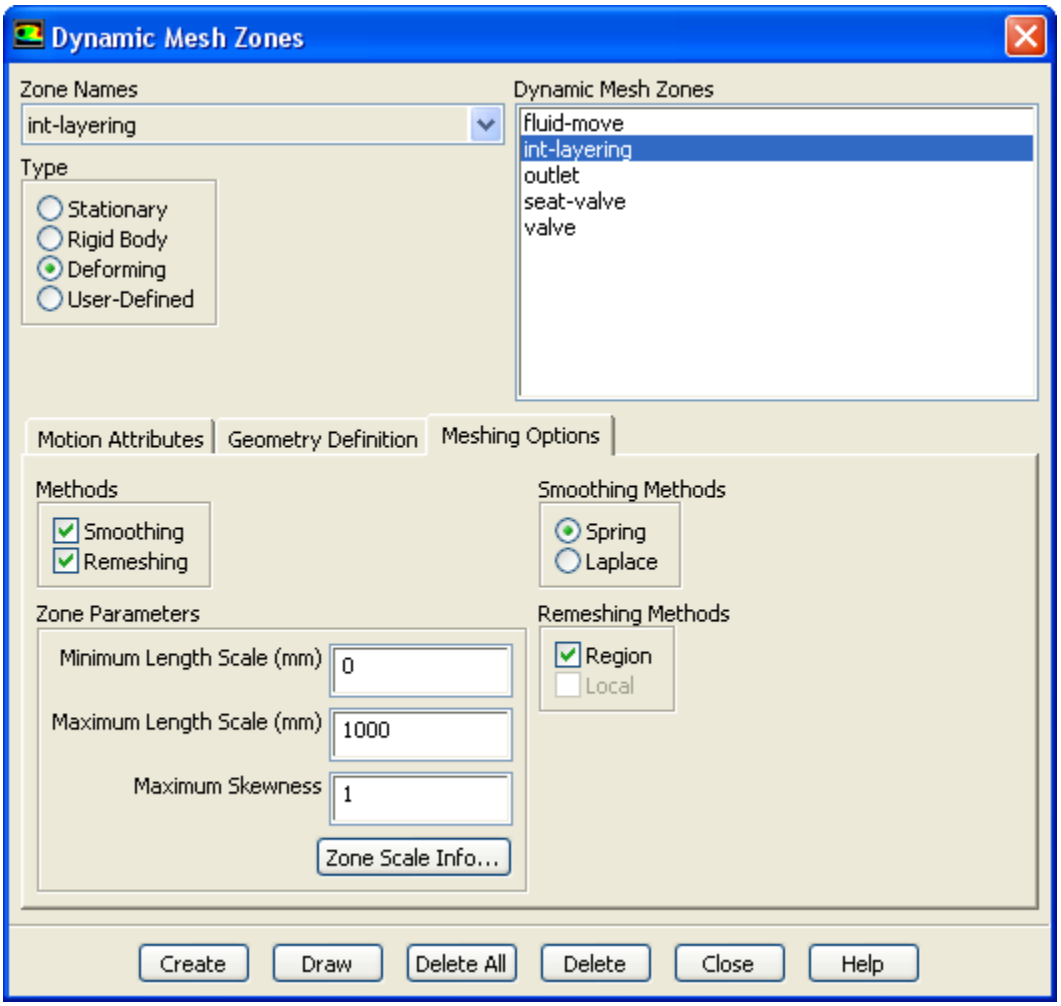
Refer to the UDF Manual for information about UDFs.

10. If the rigid body zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab, as described in step 4. of *Stationary Zones* (p. 614).
11. Click **Create**.

11.6.7.4. Deforming Motion

To define a deforming zone in your model, follow the steps below.

Figure 11.79 The Dynamic Mesh Zones Dialog Box for a Deforming Motion



1. Select the deforming zone in the **Zone Names** drop-down list.
2. Select the **Deforming** option under **Type**.
3. Specify the geometry of the deforming zone in the **Geometry Definition** tab. There are four options:
   - If no geometry is available, select **faceted** in the **Definition** drop-down list.
   - If the geometry is a plane, select **plane** in the **Definition** drop-down list. To define the plane, enter the position of a point on the plane in **Point on Plane** and the plane normal in **Plane Normal**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the geometry is a cylinder, select **cylinder** in the **Definition** drop-down list. To define the cylinder, enter the **Cylinder Radius**, the **Cylinder Origin** and the **Cylinder Axis**.
- If the geometry is described by a user-defined function, select **user-defined** in the **Definition** drop-down list and the appropriate user-defined functions in the **Geometry UDF** drop-down list. See the UDF Manual for information on user-defined functions.

For 3D simulations, ANSYS FLUENT allows you to preserve features not only between the different face zones, but also within a face zone. For any geometry definition (**faceted**, **plane**, **cylinder**, or **user-defined**), you can indicate whether you want to include features of a specific angle by selecting **Include Features** under **Feature Detection** and setting the **Feature Angle** in degrees. For more information, see *Feature Detection* (p. 586).

When available, the geometry information is used to project nodes on the deforming zone after remeshing the face zone, or if nodes are moved from the spring-based smoothing method.

4. For deforming face and cell zones that are not CutCell zones, specify the appropriate remeshing parameters in the **Meshing Options** tab.

   You can locally disable or enable **Smoothing** and or **Remeshing** and use any **Smoothing** and or **Remeshing** method.

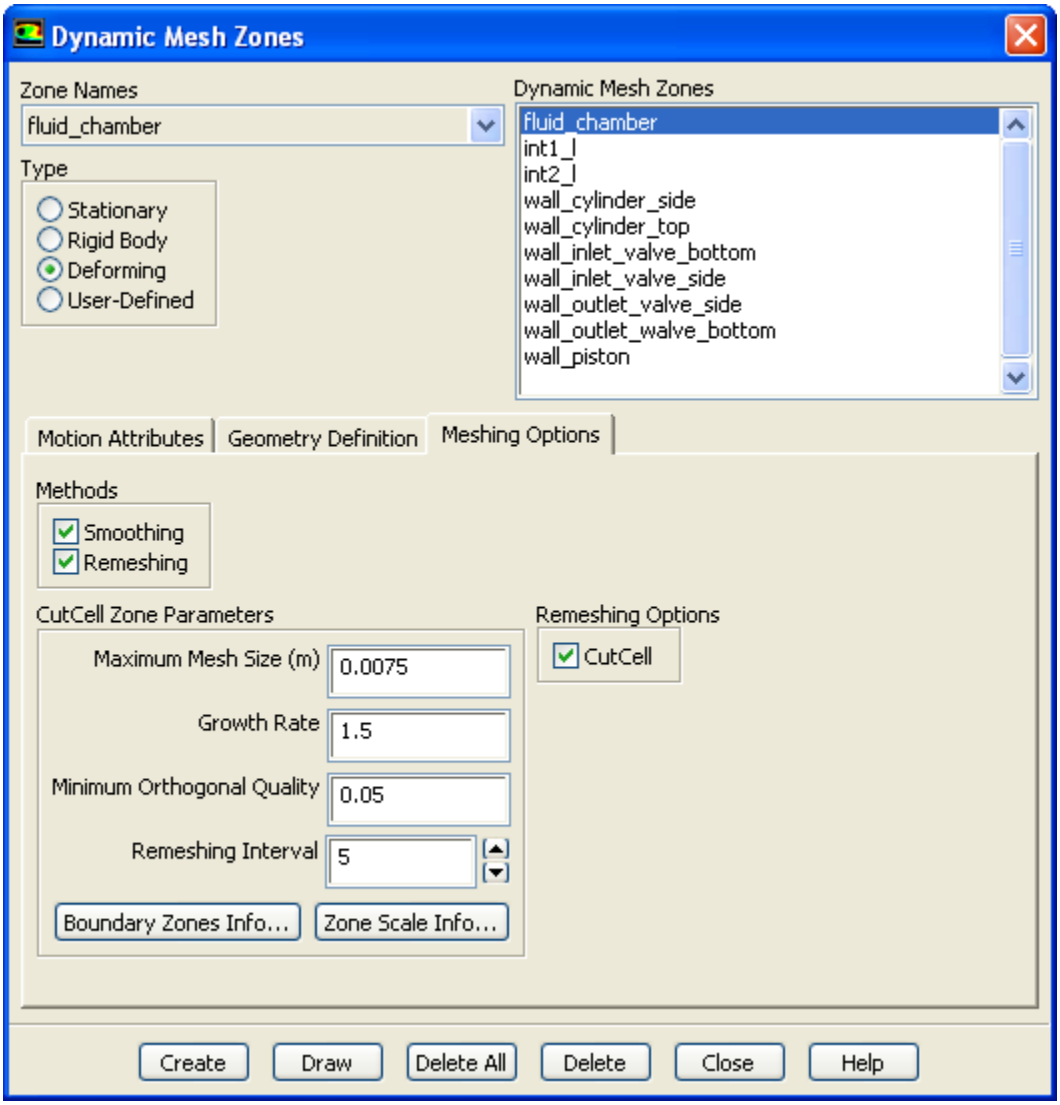
   You can view the vital statistics of your zone by clicking the **Zone Scale Info...** button. This opens the *Zone Scale Info Dialog Box* (p. 1894), where you can view the minimum and maximum length scale values, as well as the maximum skewness values.

   If you selected a cell or face zone, you need to enter **Minimum Length Scale**, **Maximum Length Scale** and **Maximum Skewness** if you want impose a different set of remeshing criteria, other than those you specified globally in the *Dynamic Mesh Task Page* (p. 1877). This is not required for cell zones since the global settings for the dynamic mesh parameters are used if ANSYS FLUENT determines that the local settings are unreasonable. You should use the information found in the *Zone Scale Info Dialog Box* (p. 1894) in order to set your values.

5. To specify that a deforming cell zone is remeshed using the CutCell zone method, enable the **CutCell** option in the **Remeshing Options** group box of the **Meshing Options** tab (see *Figure 11.80* (p. 621)). Note that this option is only available if you have previously enabled the **CutCell Zone** option in the **Remeshing** tab of the **Mesh Method Settings** dialog box (see *Figure 11.33* (p. 569)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.80 The Dynamic Mesh Zones Dialog Box for a Deforming CutCell Cell Zone



After the **CutCell** option has been enabled, you need to enter values for the global CutCell parameters **Maximum Mesh Size** and **Growth Rate**, as well as the controls for the remeshing frequency, **Minimum Orthogonal Quality** and **Remeshing Interval**.

The **Maximum Mesh Size** specified for the CutCell cell zone controls the global maximum size of the Cartesian cells. The global **Growth Rate** is used for all mesh-based size functions applied to CutCell boundary zones that are not defined as dynamic zones. When selecting values for these controls, it may be helpful to view the statistics of your cell zone by clicking on the **Zone Scale Info...** button. This opens the *Zone Scale Info Dialog Box* (p. 1894), where you can view the minimum and maximum length scale values, as well as the maximum skewness values.

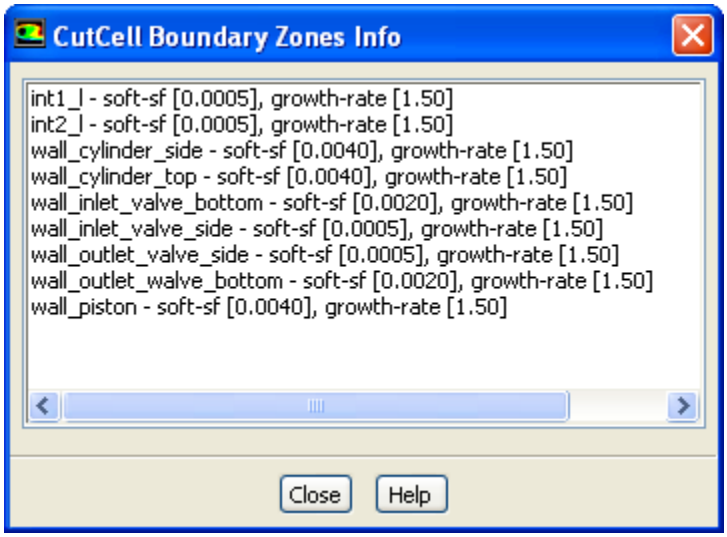
The CutCell remeshing is performed automatically whenever cells exhibit an orthogonal quality (as defined in *Mesh Quality* (p. 144)) that is less than the specified **Minimum Orthogonal Quality**. The remeshing is also performed periodically after the calculation has undergone the number of time steps you specified for the **Remeshing Interval**.

Click the **Boundary Zones Info...** button to open the **CutCell Boundary Zones Info** dialog box (*Figure 11.81* (p. 622)). This dialog box lists all the boundary zones associated with the CutCell cell zone, along with the associated size functions and zonal meshing parameters. It is recommended that you revisit

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

this dialog box after you have created all of the dynamic zones, in order to review all of the parameters you have set.

Figure 11.81 The CutCell Boundary Zones Info Dialog Box



6. If the deforming zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab, as described in step 4. of *Stationary Zones* (p. 614).
7. Click **Create**.

11.6.7.5. User-Defined Motion

For a zone that is deforming and moving, you can define the position of each node on the general deforming/moving zone using a user-defined function (UDF). To define a moving and deforming zone, follow the steps below.

1. Select the moving and deforming zone in the **Zone Names** drop-down list.
2. Select the **User-Defined** option under **Type**.
3. In the **Motion Attributes** tab, select the user-defined function that defines the geometry and motion of the zone from the **Mesh Motion UDF** drop-down list. See the UDF Manual for information on user-defined functions used to specify user-defined motion.
4. For a face zone that is not a boundary of a CutCell dynamic cell zone, you can specify the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** which is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in *Equation 11–14* (p. 565) and *Equation 11–15* (p. 566) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS FLUENT, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the separate UDF Manual for information about UDFs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If the face zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab, as described in step 4. of *Stationary Zones* (p. 614).
6. Click **Create**.

11.6.7.5.1. Specifying Boundary Layer Deformation Smoothing

For a boundary layer that deforms according to the adjacent face zone, the zone that is deforming and moving is defined using a user-defined function (UDF), as described in *User-Defined Motion* (p. 622). To define a moving and deforming boundary layer, follow the steps below:

1. Select the moving and deforming zone in the **Zone Names** drop-down list.
2. Select the **User-Defined** option under **Type**.
3. In the **Motion Attributes** tab, select the user-defined function that defines the geometry and motion of the zone from the **Mesh Motion UDF** drop-down list.
4. Click **Create**.
5. Create a deforming dynamic zone for the boundary layer fluid zone by selecting the zone in the **Zone Names** drop-down list. Note that the boundary layer has to be a separate fluid zone from the adjacent fluid zone. Select **Deforming** under **Type** and enable **Smoothing** on the boundary layer fluid zone.
6. Click **Create**.
7. Create a deforming dynamic zone for the fluid zone outside the boundary layer by selecting the appropriate zone from the **Zone Names** drop-down list. Select **Deforming** under **Type** and enable **Smoothing** and **Remeshing** in the **Methods** group box. Enabling both methods is necessary because the deforming boundary layer will deform the adjacent cells.
8. Click **Create**.

11.6.7.6. Solid-Body Kinematics

ANSYS FLUENT uses solid-body kinematics if the motion is prescribed based on the position and orientation of the center of gravity of a moving object. This is applicable to both cell and face zones.

The motion of the solid-body can be specified either as a profile or as a user-defined function (UDF). A profile may be defined by the following profile fields:

- time (time)
- crank angle (angle) (in-cylinder flows only)
- position (x, y, z)
- linear velocity (v_x, v_y, v_z)
- angular velocity (ω_x, ω_y, ω_z)
- orientation (θ_x, θ_y, θ_z)

For in-cylinder simulations, the velocity profiles for valves can be expressed as a function of crank angle instead of time. In addition, transient boundary condition profiles can also be expressed as a function of crank angle instead of time. For more information about transient profiles, see *Defining Transient Cell Zone and Boundary Conditions* (p. 376).

Below are two examples of a profile format:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
((movement_linear 3 point)
(time
0  1  2 )
(x
2  3  4 )
(v_y
0  -5  0 )
)

((movement_angular 3 point)
(time
0  1  2 )
(omega_x
2  3  4 )
)
```

For in-cylinder flows, crank angles can be included in transient tables as well as transient profiles, in a similar fashion to time. An example of a transient table using (crank) angle is as follows:

```
example 2 3 1
angle temperature
0  300
180 500
360 300
```

An example of a transient profile using (crank) angle is as follows:

```
((example transient 3 1)
(angle
0.000000e+00 1.800000e+02 3.600000e+02)
(temperature
3.000000e+02 5.000000e+02 3.000000e+02)
)
```

In addition to the motion description, you must also specify the starting location of the center of gravity and orientation of the solid body. In 2D cases (and 3D cases that do not use the six DOF solver), ANSYS FLUENT automatically updates the center of gravity position and orientation at every time step such that

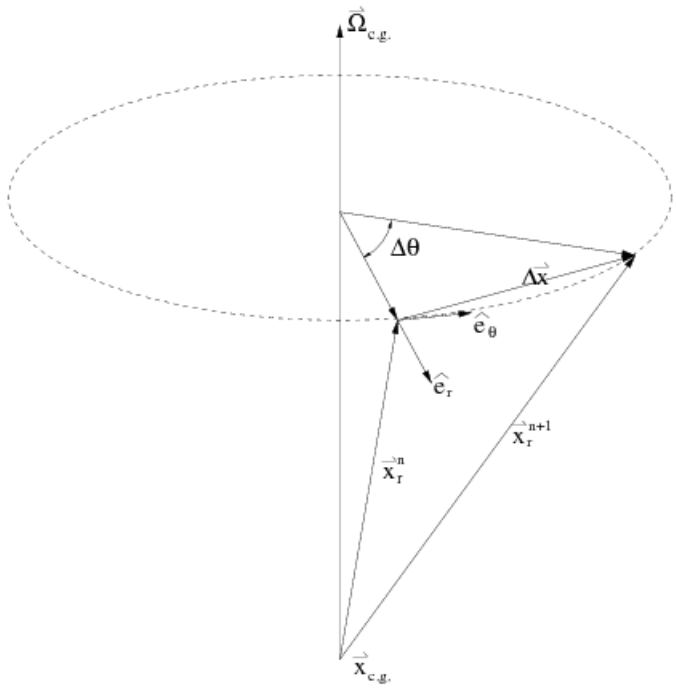
$$\begin{aligned} \vec{x}_{c.g.}^{\,n+1} &= \vec{x}_{c.g.}^{\,n} + \vec{v}_{c.g.}\Delta t \\ \vec{\theta}_{c.g.}^{\,n+1} &= \vec{\theta}_{c.g.}^{\,n} + \vec{\Omega}_{c.g.}\Delta t \end{aligned} \qquad \textbf{(11–31)}$$

where $\vec{x}_{c.g.}$ and $\vec{\theta}_{c.g.}$ are the position and orientation of the center of gravity, $\vec{v}_{c.g.}$ and $\vec{\Omega}_{c.g.}$ are the linear and angular velocities of the center of gravity. 3D, six DOF cases use a more complex form of *Equation 11–31* (p. 624) when updating θ.

Typically, $\vec{\theta}$ is chosen to be an appropriate set of Euler angles. In this case, the solid-body motion must be specified using a user-defined function (`DEFINE_CG_MOTION`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.82 Solid Body Rotation Coordinates



The position vectors on the solid body are updated based on rotation about the instantaneous angular velocity vector $\vec{\Omega}_{c.g.}$. For a finite rotation angle $\Delta\theta = \left|\vec{\Omega}_{c.g.}\right|\Delta t$, the final position of a vector $\vec{x}_r$ on the solid body with respect to $\vec{x}_{c.g.}$ can be expressed as (See *Figure 11.82* (p. 625))

$$\vec{x}_r^{n+1} = \vec{x}_r^{n} + \Delta\vec{x} \tag{11–32}$$

where $\Delta\vec{x}$ can be shown to be

$$\Delta\vec{x} = \left|\vec{x}_r^{n} - \vec{x}_{c.g.}\right|[\sin(\Delta\theta)\hat{e}_\theta + (\cos(\Delta\theta) - 1)\hat{e}_r] \tag{11–33}$$

The unit vectors $\hat{e}_\theta$ and $\vec{e}_r$ are defined as

$$\hat{e}_\theta = \frac{\vec{\Omega}_{c.g.} \times \vec{x}_r}{\left|\vec{\Omega}_{c.g.} \times \vec{x}_r\right|} \tag{11–34}$$

$$\hat{e}_r = \frac{\hat{e}_\theta \times \vec{\Omega}_{c.g.}}{\left|\hat{e}_\theta \times \vec{\Omega}_{c.g.}\right|} \tag{11–35}$$

If the solid body is also translating with $\vec{v}_{c.g.}$, the $n+1$ position vector on the solid body can be expressed as

$$\vec{x}^{\,n+1} = \vec{x}^{\,n}_{c.g.} + \vec{v}_{c.g.}\Delta t + \vec{x}^{\,n+1}_{r} \qquad \textbf{(11–36)}$$

where $\vec{x}^{\,n+1}_{r}$ is given by *Equation 11–32* (p. 625).

11.6.8. Previewing the Dynamic Mesh

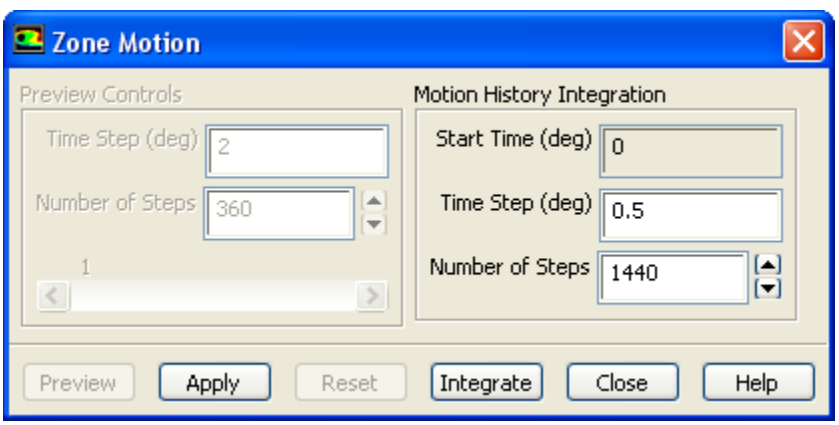
When you have specified the mesh update methods and their associated parameters, and you have defined the motion of dynamic zones, as described in *Specifying the Motion of Dynamic Zones* (p. 613), you can preview the motion of the mesh or the zone as it changes with time before you start your simulation. The same dynamic zone or mesh motion will be executed when you start your simulation.

11.6.8.1. Previewing Zone Motion

You can preview the motion of zones with rigid body motion using the *Zone Motion Dialog Box* (p. 1895) (*Figure 11.83* (p. 626)).

Dynamic Mesh → **Display Zone Motion...**

Figure 11.83 The Zone Motion Dialog Box



The zone motion preview only updates the graphical representation (in the graphics window) of the zones that you have selected using the *Mesh Display Dialog Box* (p. 1635). The zone motion preview will only update those zones that have solid body motion specified. To use the **Zone Motion** preview:

1. Select the appropriate zones to display in the *Mesh Display Dialog Box* (p. 1635).
2. In the *Zone Motion Dialog Box* (p. 1895), enter the **Time Step** and the **Number of Steps** under **Motion History Integration**.
3. Click the **Integrate** button. This allows ANSYS FLUENT to create a table of surface positions in time.
4. Under **Preview Controls**, specify the **Time Step** and the **Number of Steps** for preview. Note that the time step here can be larger than the integration time step.
5. Click **Preview** to preview the zone motion. Click **Apply** to save your settings for zone motion. Click **Reset** to have the default inputs restored in the dialog box.

You can also use the slider bar on the *Zone Motion Dialog Box* (p. 1895) to fast-forward or rewind the motion of the selected zones. Previewing the zone motion can also be used as a postprocessor for six DOF simulations (see *Using the Six DOF Solver* (p. 602)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

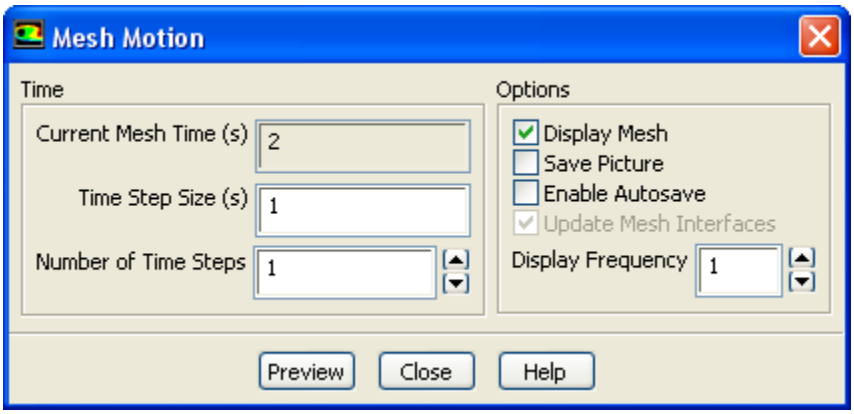
11.6.8.2. Previewing Mesh Motion

The mesh motion preview is different from the zone motion described above in that the mesh connectivity is changed in mesh motion.

To preview the dynamic mesh of a transient case, you can use the *Mesh Motion Dialog Box* (p. 1896) (*Figure 11.84* (p. 627))

Dynamic Mesh → **Preview Mesh Motion...**

Figure 11.84 The Mesh Motion Dialog Box



The procedure is as follows:

1. Save the case file.

   File → Write → **Case...**

   Important

   Note that the mesh motion will actually update the node locations as well as the connectivity of the mesh, so you must be sure to save your case file before doing the dynamic mesh motion. Once you have advanced the mesh by a certain number of time steps, you will not be able to recover the previous status of the mesh, other than by reloading the appropriate ANSYS FLUENT case file.

2. Specify the **Number of Time Steps** and the size of each time step (**Time Step Size**). The current time will be displayed in the **Current Mesh Time** field after the dynamic mesh has been advanced the specified number of steps.

   Note that if you turned on the in-cylinder option, the **Time Step Size** is automatically calculated from the **Crank Angle Step Size** and the **Crank Shaft Speed** that you have specified in the **In-Cylinder** tab of the *Options Dialog Box* (p. 1883).

3. To view the dynamic mesh in the graphics window, enable the **Display Mesh** option. In addition, you can control the frequency at which ANSYS FLUENT should display an updated mesh in the **Display Frequency** field. To save a picture file of the mesh each time ANSYS FLUENT updates it during the preview, turn on the **Save Picture** option. This opens the *Save Picture Dialog Box* (p. 1988) (see *Saving Picture Files* (p. 117)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Turn on **Enable Autosave** to use the automatic saving feature to specify the file name and frequency with which case and data files should be saved during the solution process. This opens the *Autosave Case During Mesh Motion Preview Dialog Box* (p. 1897).

   See *Automatic Saving of Case and Data Files* (p. 65) for details about the use of this feature. This provides a convenient way for you to save results at successive time steps for later postprocessing.
5. Enable the **Update Mesh Interfaces** option to update the interface at every time step.
6. Click **Preview** to start the preview. ANSYS FLUENT will update the dynamic mesh by moving and deforming the face and cell zones that you have specified as dynamic zones. Click **Apply** to save your settings for mesh motion.

During the preview, information about the dynamic mesh will be displayed in the console window for each time step. Note that for the in-cylinder option, the reported `Maximum Cell Skew` is calculated only from zones undergoing remeshing. This ensures that you can always ascertain whether the skewness is increasing in the deforming zones. To report the maximum skewness of a cell from *any* zone, you can click the **Report Quality** button in the *General Task Page* (p. 1631).

General → **Report Quality**

11.6.9. Steady-State Dynamic Mesh Applications

While many dynamic mesh problems are transient, you can use dynamic meshes for steady-state applications as well. Some examples of steady-state applications include: checking the valve application after reaching a steady-state valve position; or after a fluid-structure interface application has reached a steady-state solution.

There are no differences in the meshing aspect between steady-state cases and transient cases. Furthermore, setting up a steady-state simulation is similar to setting up a transient case, described in *Using Dynamic Meshes* (p. 548). However, there are a few differences which you should note:

- A `CG_MOTION` UDF is needed to specify the motion of the boundary: a transient profile used in transient cases cannot be used in steady-state cases.
- The `dtime` passed to the `CG_MOTION` UDF is 1 by default: if a displacement of 1 mm is needed to move the boundary, you can specify the velocity to be 1e-3 m/s.
- Dynamic mesh parameters can be different since an interpolation error is no longer a concern.
- If you have enabled local remeshing for your steady-state application, you can instruct ANSYS FLUENT to perform additional remeshing after the boundary has moved. This additional remeshing is based on skewness criteria, and can further increase the quality of your mesh. See *Dynamic Mesh Update Methods* (p. 551) for further details.

The mesh must be manually updated through journal files or execute commands. To update the mesh, you can use the *Mesh Motion Dialog Box* (p. 1896).

Run Calculation → **Preview Mesh Motion...**

Alternatively, you can use the following text command:

```
solve → mesh-motion
```

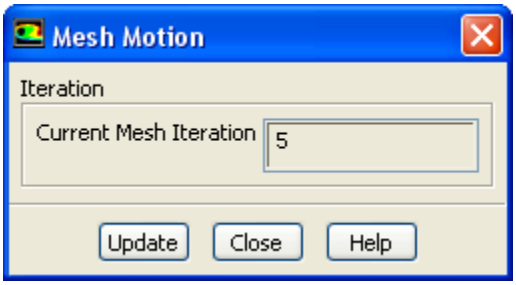
which can also be used as an execute command in the *Execute Commands Dialog Box* (p. 1950):

Calculation Activities → **Create/Edit... (Execute Commands)**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can display dynamic mesh statistics (such as minimum and maximum volumes and maximum cell and face skewness) by clicking the **Update** button in the *Mesh Motion Dialog Box* (p. 1896) (*Figure 11.85* (p. 629)).

Figure 11.85 The Mesh Motion Dialog Box for Steady-State Dynamic Meshes



Important

The following options are not available for steady-state applications:

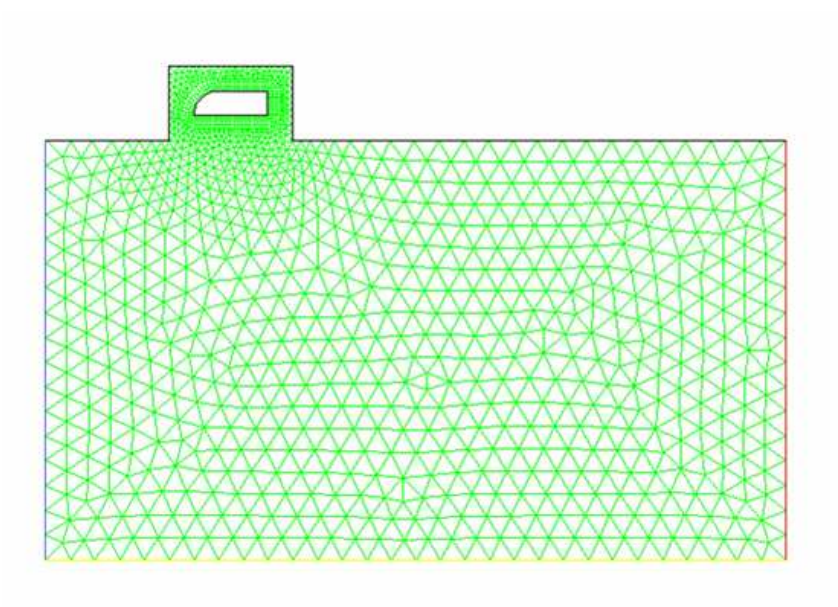
- **In-Cylinder**
- **Six DOF**
- **Implicit Update**

Also, the **Dynamic Mesh Events** dialog box is not available for steady-state applications.

11.6.9.1. An Example of Steady-State Dynamic Mesh Usage

Consider a rescue drop case shown in *Figure 11.86* (p. 629). The object can be moved in any position in the steady-state solver, after which steady-state analyses can be performed at different object positions.

Figure 11.86 Initial Object Position



The dynamic mesh parameters setup is identical for the steady-state and transient cases, which is described in *Setting Dynamic Mesh Modeling Parameters* (p. 550). When setting up the dynamic zones, the procedures

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are similar to those described in *Specifying the Motion of Dynamic Zones* (p. 613), except that the UDF selected from the **Motion UDF/Profile** drop-down list is different. In steady-state cases the `dtime` passed to the UDF is by default 1. So, in this example, the object will move 50 mm each time the following UDF is executed:

```
#include "udf.h"

DEFINE_CG_MOTION(pod,dt,vel,omega,time,dtime)

{
 NV_S(vel,=,0);
 NV_S(omega,=,0);
 vel[1] = -50e-3;
}
```

The resulting mesh is shown in *Figure 11.88* (p. 630).

Figure 11.87 The Mesh Motion Dialog Box After 40 Updates

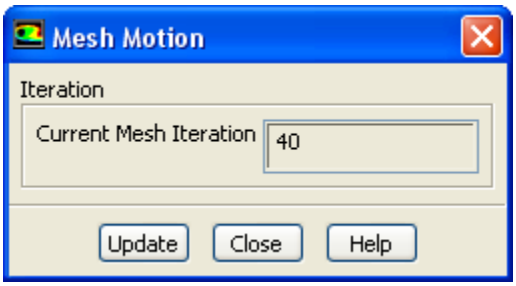
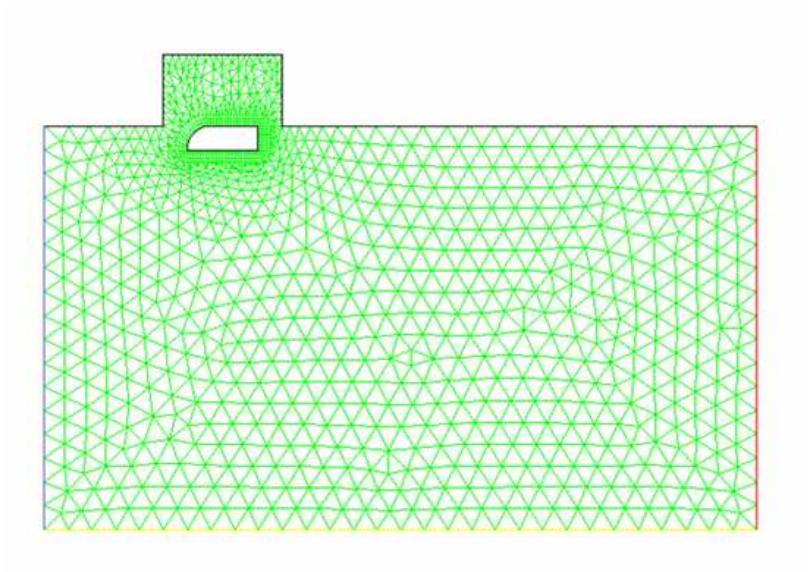


Figure 11.88 Final Object Position After 40 Executions



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: Modeling Flows Using the Mesh Morpher/Optimizer

This chapter describes the setup and use of the mesh morpher/optimizer in ANSYS FLUENT.

Details about the mesh morpher/optimizer are presented in the following sections:

12.1. Introduction
12.2. The Optimization Process
12.3. Optimizers
12.4. Setting Up the Mesh Morpher/Optimizer

12.1. Introduction

Aerodynamic shape optimization has increasingly become a critical field in the area of CFD. ANSYS FLUENT provides a mesh morphing capability that allows you to solve shape optimization problems [*37*] (p. 2211). This capability is suitable for problems that require minimal design changes in the geometry, in order to satisfy certain criteria. For example, you could optimize the shape of an aircraft wing airfoil in order to meet a specified pressure distribution that yields an objective function, such as increased range, decreased drag, or increased lift. Another example would be the optimization of an automobile exterior that minimizes drag.

The mesh morpher/optimizer in ANSYS FLUENT has the following advantages:

- The optimization problem can be solved inside ANSYS FLUENT. You can maintain the topology of the initial design, while generating solutions for desired configurations without having to go back and change the geometry and mesh it again.
- The mesh morpher can handle multiple deformation regions.

For additional information, please see the following section:

12.1.1. Limitations

12.1.1. Limitations

Note the following limitations of the ANSYS FLUENT mesh morpher/optimizer:

- Arbitrarily-shaped deformation regions are not supported.
- If the surface to surface (S2S) radiation model is enabled, the view factors will need to be recomputed at the very least when the optimization is complete. If the objective function is a function of temperature, the view factors will need to be recomputed after each design change.
- The mesh morpher/optimizer should not be used with dynamic or sliding mesh problems.
- If you restart a case file with a mesh that was deformed by the built-in optimizer, you must revise the mesh morpher/optimizer settings for the revised mesh.

12.2. The Optimization Process

All optimization problems require that you identify parameters that can be modified in order to reach the optimized solution. In the case of the mesh morpher/optimizer, it is the geometry that needs to be parameterized. Geometric parameterization for general shapes used in CFD can be very complicated, due to the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

large variety of shapes available in engineering applications. In order to minimize such complications in your ANSYS FLUENT simulation, the problem of shape parameterization is reduced to a problem of the parameterization of changes in the geometry.

The next essential requirement for mesh morphing is a tool that can smoothly alter the shape, irrespective of the underlying mesh topology. In ANSYS FLUENT, this is accomplished using a free form deformation technique, which manipulates designated deformation regions and control points. The mesh region that is to be deformed is overlaid with a structured multi-block grid, comprised of control points. The control points are defined in terms of global coordinates. The tensor product of Bernstein polynomials are used to define the smooth deformation. The parameterization of the shape is then achieved by changing the location of the control points.

The next requirement is to have an optimizer that is robust enough to handle a wide range of problems. By coupling such optimizers with your CFD analysis, you can greatly improve your design with minimal intervention. You can use built-in optimizers to satisfy the condition as specified through an objective function. The mesh morpher/optimizer provides you with access to five optimizers that are not based on gradients. Alternatively, you can manually specify the values of the parameters, and then run the case with the deformed mesh to see if a given condition is satisfied.

12.3. Optimizers

The internal optimizers used as part of the mesh morpher/optimizer capability in ANSYS FLUENT use direct search methods for optimization. Direct search methods are zeroth order, as they only use the objective function values for optimization. The direct search methods do not use the derivatives.

The following is a list of advantages of direct search methods:

- Direct search methods do not require derivatives for optimization.
- Direct search methods are robust for problems with discontinuities and in situations where the derivative computation is not possible or unreliable.

The following is a list of disadvantages of direct search methods:

- Convergence proof is not clearly defined.
- The rate of convergence can be very slow.

General explanations of the five different internal optimizers available in ANSYS FLUENT are provided in the following sections:

12.3.1. The Compass Optimizer
12.3.2. The Simplex Optimizer
12.3.3. The Torczon Optimizer
12.3.4. The Powell Optimizer
12.3.5. The Rosenbrock Optimizer

12.3.1. The Compass Optimizer

In the Compass optimizer [*110* (p. 2214)], the parameters are adjusted one by one until the objective function is minimized. This optimizer starts with a given value and then evaluates the function value in all the basic directions. The direction here refers to the positive and negative increments to the initial parameter values. If there is a reduction in the function value, then that point becomes an improved point. If there is no improvement in the function values, then the step length is reduced by half and the search is repeated in all directions. The algorithm terminates when the step size falls below a certain tolerance.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Compass optimizer initially makes rapid progress towards the solution. While this method might quickly approach the minimum value of an objective function, it may be slow to detect this fact. It may also converge very slowly if the level sets of the objective function are extremely elongated.

12.3.2. The Simplex Optimizer

The Simplex optimizer is also referred to as the Downhill Simplex optimizer [*111* (p. 2214)], [*113* (p. 2215)] and the Nelder-Mead method [*112* (p. 2215)]. It is based on the idea of geometric simplexes; for example, a 2D simplex is a triangle, and a 3D simplex is a tetrahedron.

For optimization purposes, ANSYS FLUENT requires that simplexes are regular polyhedra (i.e., not degenerate polyhedra with collapsed sides). Each vertex of the geometric simplex represents one function evaluation (which in this case is one CFD run), and the number of vertices corresponds to the number of parameters. For the free form deformation method that is used by ANSYS FLUENT to parameterize changes in shapes, the number of active control points will determine the number of vertices of the geometric simplex.

Minimization of the objective function is performed based on a set of the rules about the "quality" of each vertex. The vertex quality is the value of the function evaluated for each position of the control point. A set of geometric operations such as reflection, expansion, contraction, and shrinking are performed in order to find the region in which to look for the minimum of the function. Because optimization here is formulated as a minimization problem, the simplex optimizer algorithm seeks the "worst" vertex, that is, the vertex that has the largest value when the corresponding parameter is evaluated. By performing the reflection around the center of the gravity, the new value of the function is obtained after performing the CFD run. Similarly, the operations of expansion, contraction, and shrinking are used to obtain the minimum of the function.

The Simplex optimizer is known to work well, but it suffers from the large number of function evaluations. It also requires smooth objective functions for convergence.

12.3.3. The Torczon Optimizer

The Torczon optimizer [*107* (p. 2214)] is a slightly modified version of the simplex optimizer described previously. Given an initial vertex, this optimizer tries to find a better vertex that has a function value that is strictly less than the function value at the previous best vertex. There are three possible trial steps: the rotation step, the expansion step, and the contraction step. The algorithm always computes the rotation step and then tests to see if a new best vertex has been identified. If it has, then the expansion step is computed. Otherwise, the algorithm computes and automatically accepts the contraction step.

12.3.4. The Powell Optimizer

For the Powell optimizer, the method used is based on the number of dimensions of the problem. For optimization problems of single dimension (i.e., problems with a single parameter), the golden section search algorithm [*108* (p. 2214)] is used to find the optimal value for the objective function. In this algorithm, the optimal value is found by reducing the size of the bracketing triplet, until the size of the bracket (i.e., the distance between the outer points of the triplet) reaches a certain tolerance level. In all the cases, the middle point of the new triplet is determined to be the best value obtained so far.

For multidimensional problems (i.e., problems with multiple parameters), the minimum value and the largest decrease is found using the golden search algorithm for each dimension, in order to find the conjugate directions. A conjugate direction is a direction that when searched will not alter the minimum value attained by the previous movement in another direction; in other words, a direction in which the gradient is perpendicular to the first direction. After finding the N linearly independent, mutually conjugate directions, one pass will find the exact minimum value. For functions that are not quadratic, repeated cycles of N line minimizations will converge to minimum.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.5. The Rosenbrock Optimizer

In the Rosenbrock optimizer [*109* (p. 2214)], after the initial direction is found, multiple steps are taken in that direction until the least value is attained. The process starts with an arbitrary length e. If this initial step succeeds (i.e., the new value of the function is less than or equal to the old value), the length e is multiplied by α, where α is more than 1. If the steps fails, the length e is multiplied by β, where β is between -1 and 0. The direction or the parameter which needs to be modified is determined by advancing all the parameters by the step length e and then selecting the best among those which yield a function value that is less than the previous value. After that point is accepted as the best point, the process is repeated. These steps continue to repeat until e becomes so small so that any further change in the value of e does not significantly reduce the value of the function.

12.4. Setting Up the Mesh Morpher/Optimizer

The procedure for setting up and using the mesh morpher/optimizer for shape optimization is as follows:

1. Read the case into ANSYS FLUENT.

   File → **Read** → **Case...**

2. If you want to use one of the built-in optimizers rather than specifying the deformation manually, you will need to provide an objective function in one of two ways: either as a UDF, or as a Scheme source file. The goal of the optimizer is to deform the mesh in such a way that this objective function is minimized. ANSYS FLUENT will run the solution for a given design stage until convergence is reached, and then check to see if the objective function is satisfied. If the objective function is not satisfied, ANSYS FLUENT will proceed to the next design, and so on, until convergence is achieved from the point of view of the optimizer. You can specify the optimizer convergence criteria in step 7.d.iii.

   a. If you want to provide the objective function as a user-defined function (UDF), perform the steps that follow. For more information about UDFs, see the separate UDF Manual.

      i. Write a UDF using the `DEFINE_ON_DEMAND` macro to define the objective function. The function name must be `objective_function`. At the end of the objective function, the rpvar `morpher/objective_function` needs to be set to the `(current - target)` value. It is this value that the optimizer will attempt to minimize.

      ii. Compile the UDF using the **Compiled UDFs** dialog box. Make sure `libudf` appears as the **Library Name**.

      Define → **User-Defined** → **Functions** → **Compiled...**

   b. If you want to provide the objective function as a Scheme source file, perform the following steps:

      i. Write a Scheme source file to define the objective function. The procedure name must be `objective-function`. At the end of the objective function, the rpvar `morpher/objective-function` needs to be set to the `(current - target)` value. It is this value that the optimizer will attempt to minimize.

      ii. Load the Scheme source file (see *Reading Scheme Source Files* (p. 73) for details).

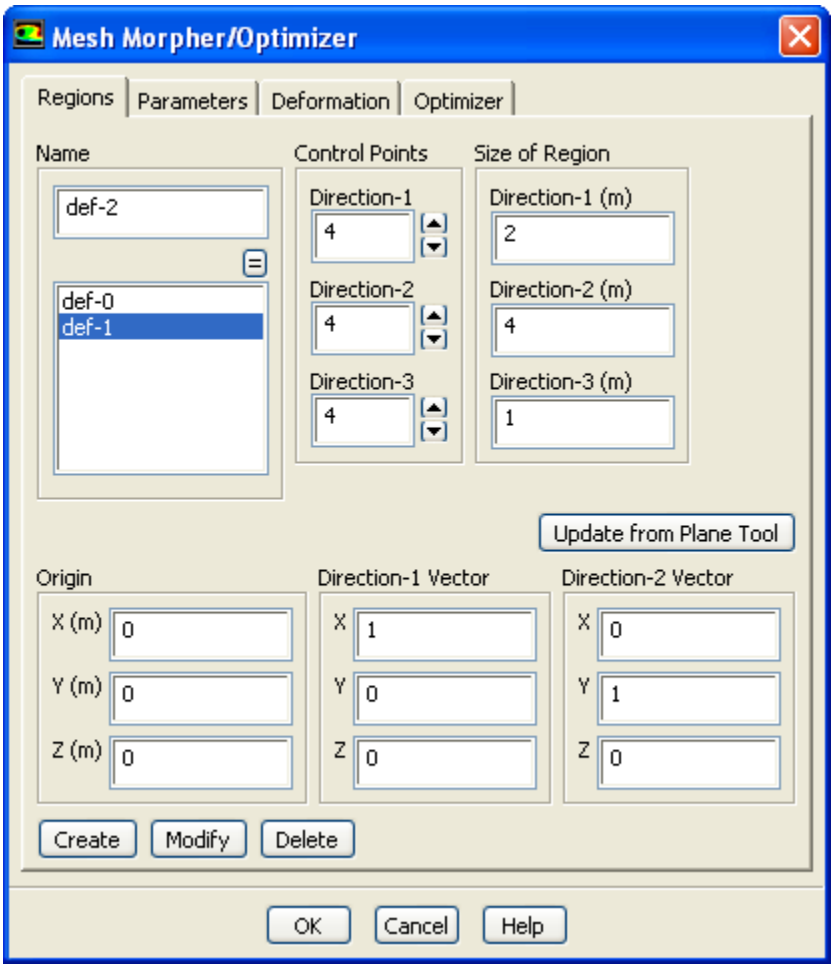
3. Open the **Mesh Morpher/Optimizer** dialog box by clicking the **Define/Mesh Morpher/Optimizer...** menu item.

   Define → **Mesh Morpher/Optimizer...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The first time you select the **Define/Mesh Morpher/Optimizer...** menu item in a session, a question dialog box will ask you whether you want to enable the mesh morpher/optimizer. Click **Yes** to load the libraries and open the **Mesh Morpher/Optimizer** dialog box (*Figure 12.1* (p. 635)).

Figure 12.1 The Regions Tab of the Mesh Morpher/Optimizer Dialog Box



4. Define the region(s) of the domain where the mesh will be deformed in order to optimize the shape, by performing the following steps in the **Mesh Morpher/Optimizer** dialog box. Each deformation region will be defined as a "box", that is, a rectangle for 2D cases and a rectangular hexahedron for 3D cases.

Important

It is recommended that you define each deformation region to be bigger than the area of interest, in order to maintain proper continuity between deforming and non-deforming regions.

a. Click the **Regions** tab to reveal the settings shown in *Figure 12.1* (p. 635).

b. Enter a name for a deformation region in the text-entry box at the top of the **Name** group box.

c. Define an origin for the deformation region by entering the Cartesian coordinates of a point in the **X**, **Y**, and (for 3D) **Z** number-entry boxes in the **Origin** group box.

d. Define the first direction vector of the deformation region relative to the **Origin** coordinates, by entering values in the **X**, **Y**, and (for 3D) **Z** number-entry boxes in the **Direction-1 Vector** group box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For 2D cases, ANSYS FLUENT will automatically define the second direction vector of the deformation region to be perpendicular to the **Direction-1 Vector**, and will display the components in the uneditable **Direction-2 Vector** group box.

e. For 3D cases, define the second direction vector of the deformation region relative to the **Origin** coordinates, by entering values in the **X**, **Y**, and **Z** number-entry boxes in the **Direction-2 Vector** group box. If the vector you define is not perpendicular to the **Direction-1 Vector**, ANSYS FLUENT will automatically redefine the second vector to be the projection of the **Direction-2 Vector** you entered onto a plane that is perpendicular to the **Direction-1 Vector**. Based on this definition, the **Direction-2 Vector** you define cannot be colinear with the **Direction-1 Vector**.

   The third direction vector of the deformation region will be automatically defined as the cross product of the first and second direction vectors.

f. You have the option of using the line tool (for 2D cases) or the plane tool (for 3D cases) to define the direction vectors of the deformation region. Set up the line tool or the plane tool (as described in *Using the Line Tool* (p. 1363) and *Using the Plane Tool* (p. 1367), respectively), and then click the **Update from Line Tool** or **Update from Plane Tool** button to update the values in the **Direction-1 Vector** and (for 3D cases) **Direction-2 Vector** group boxes.

g. Define the overall dimensions of the deformation region by entering length values for **Direction-1**, **Direction-2**, and (for 3D cases) **Direction-3** in the **Size of Region** group box.

h. Define the number of control points into which you want to divide the deformation region by entering values for each axis (**Direction-1**, **Direction-2**, and (for 3D cases) **Direction-3**) in the **Control Points** group box. The total number of control points for the region will be the product of the numbers you enter. Increasing the number of control points allows you greater control of the deformation, but also increases the computational expense.

Important

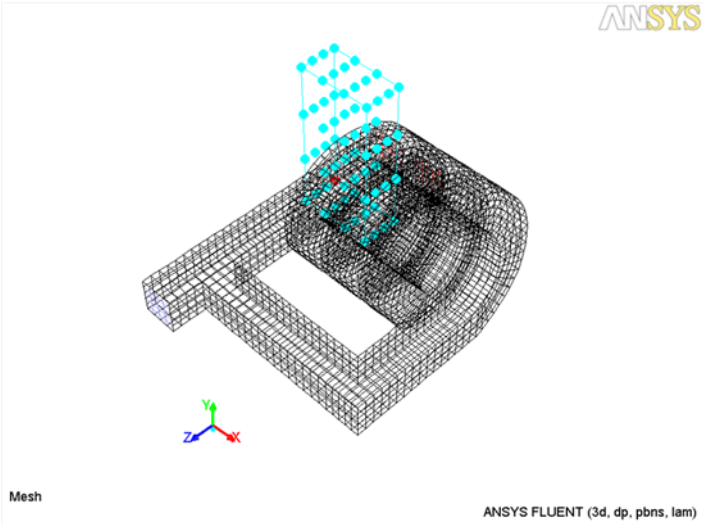
When you define multiple deformation regions, you must ensure that all the deformation regions have the same number of control points in each direction.

i. Save the settings you have created for the deformation region by clicking the **Create** button. The name of the deformation region will be added (and selected) in the selection list at the bottom of the **Name** group box. A new default name will be automatically entered in the text-entry box at the top of the **Name** group box, in preparation for any deformation regions you wish to create in the future.

   Note that the control points of the deformation region selected in the **Name** selection list will be automatically displayed in the graphics window (see *Figure 12.2* (p. 637)). If you would like to have no control points displayed in the graphics window, you can deselect the item by clicking the button located above the right side of the **Name** selection list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.2 Displaying the Control Points of a Deformation Region



j. If you need to modify the deformation region at any point, select it in the **Name** selection list, revise the appropriate settings, and click the **Modify** button. The updated control points will be displayed in the graphics window.

Important

When modifying an existing deformation region, be sure to click **Modify** rather than **Create**. If you click **Create**, you will not modify the deformation region, but will instead create a new one with the modified settings.

k. If you need to delete the deformation region at any point, select it in the **Name** selection list and click the **Delete** button.

l. Create any additional deformation regions as necessary by repeating steps 4.b.-k.

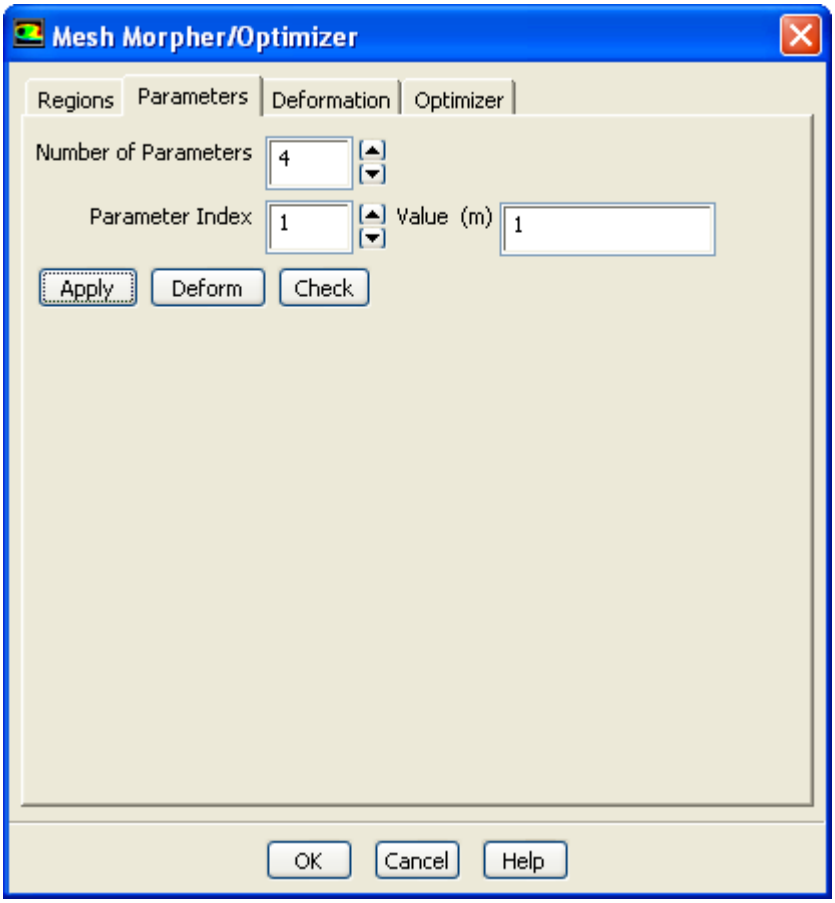
Important

Overlapping deformation regions are not supported.

5. Set up the parameters for deformation.

   a. Click the **Parameters** tab to reveal the settings shown in *Figure 12.3* (p. 638).

Figure 12.3 The Parameters Tab of the Mesh Morpher/Optimizer Dialog Box



b. Enter the **Number of Parameters**. This number may be as low as the maximum number of parameters you will define on a single control point, or as high as the total number of parameters you will set on all of the control points combined.

c. If you want to manually specify the deformation rather than using the built-in optimizers, perform the following steps to define the parameters:

   i. Click the **Optimizer** tab to reveal the settings shown in *Figure 12.5* (p. 641).

   ii. Select **none** from the **Optimizer** drop-down list.

   iii. Click the **Parameters** tab.

   iv. Enter the **Parameter Index** for the parameter you wish to define. Note that the first parameter has an index of **0**.

   v. Enter the **Value** for the parameter specified in **Parameter Index**. The **Value** defines a magnitude of deformation for the parameter, which is then multiplied with each of the directional components you specify in the **Scaling Factors** group box of the **Deformation** tab to produce the overall displacement of a control point.

      Note that the **Value** text-entry box is only editable when you have **none** selected from the **Optimizer** drop-down list in the **Optimizer** tab.

   vi. Click the **Apply** button to save the setting for the parameter.

   vii. At any point you can use the **Deform** and **Check** buttons, to display the deformed mesh in the graphics window and to check the mesh, respectively. These buttons are more fully de-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

scribed in the steps that follow, as they are also available in the **Deformation** tab and are more useful after you have finalized the **Deformation** tab settings.

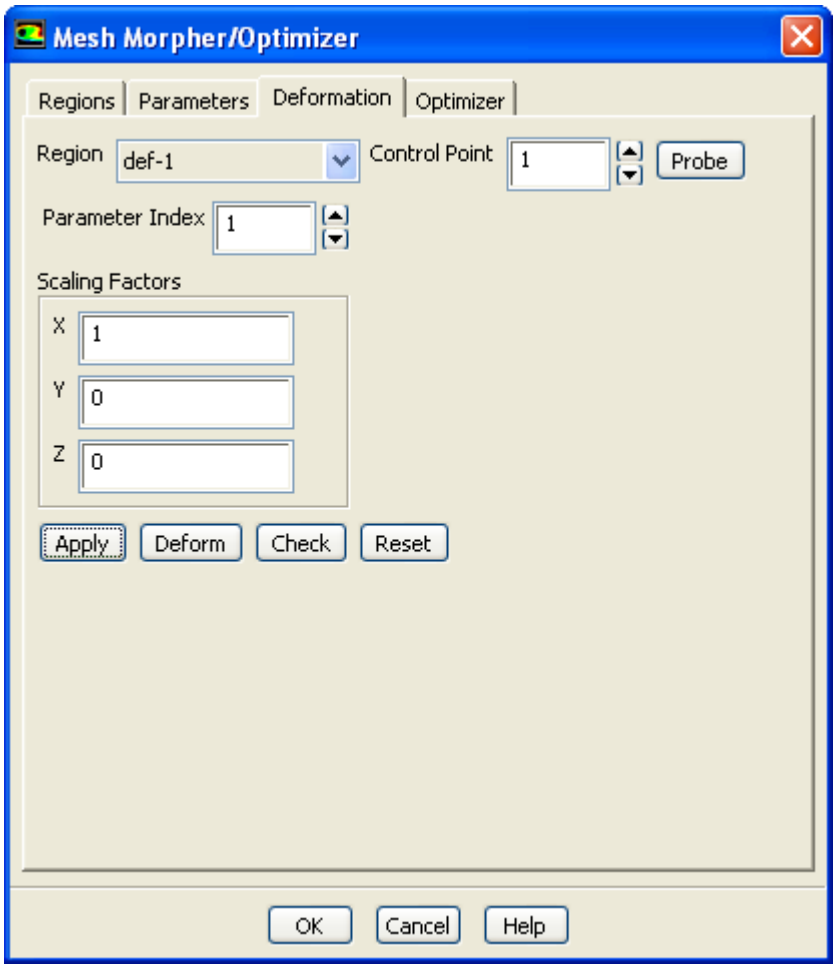
viii. Repeat steps 5.c.iv.-vii. until all of the parameters have been defined.

6. Assign the deformation parameters to the control points.

   a. Click the **Regions** tab and select a region from the **Name** selection list, so that it is displayed in the graphics window.

   b. Click the **Deformation** tab to reveal the settings shown in *Figure 12.4* (p. 639).

Figure 12.4 The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box



   c. Select the same region you selected in step 6.a. (i.e., the displayed region) from the **Region** drop-down list.

   d. Specify a control point in the current **Region** to which you want to assign deformation parameters by entering the index number for **Control Point**. The current **Control Point** will be highlighted in the graphics window; if you want to see the highlighted control point and cannot locate it, you should adjust the view in the graphic display window.

      Alternatively, you can specify the control point by clicking the **Probe** button and clicking on a control point in the graphics window with the **mouse-probe** button of the mouse (see *Controlling the Mouse Button Functions* (p. 1427) for details).

   e. Enter the **Parameter Index** of a parameter you want to assign to the current **Control Point**. Note that the first parameter has an index of **0**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

f. Define the scaling factors that will be applied to the deformation parameters in the **X**, **Y**, and (for 3D) **Z** directions by entering values in the **Scaling Factors** group box. If you use an built-in optimizer, these scaling factors provide the direction of the displacement of the control point and the optimizer determines the overall magnitude of displacement. Alternatively, if you manually specify the deformation, the values you enter in the **Scaling Factors** group box are multiplied with the **Value** specified in the **Parameters** tab to define the displacement of the control point.

g. Click the **Apply** button to save the settings for the current **Parameter Index** of the current **Control Point**.

h. If **none** is selected from the **Optimizer** selection list in the **Optimizer** tab, you can click the **Deform** button to display the deformed mesh (that results from the saved parameter settings) in the graphics display. The **Deform** button allows you to manually specify the deformation.

i. If **none** is selected from the **Optimizer** selection list in the **Optimizer** tab, you can click the **Check** button to print out a mesh check report in the console for the currently displayed mesh. The mesh check report is the same as that produced by the **Check** button in the **General** task page, as described in *Mesh Check Report* (p. 173).

Important

If you save parameter settings but do not click the **Deform** button, the mesh check report will not account for the mesh that is produced from those settings.

---

j. Click the **Reset** button if you want to revert to the original mesh without the deformations that result from the **Deform** button.

k. Repeat steps 6.e.-j. for each additional parameter you want to define for the current **Control Point** in the current **Region**.

l. Repeat steps 6.d.-k. for each additional control point in the current **Region** to which you want to assign deformation parameters.

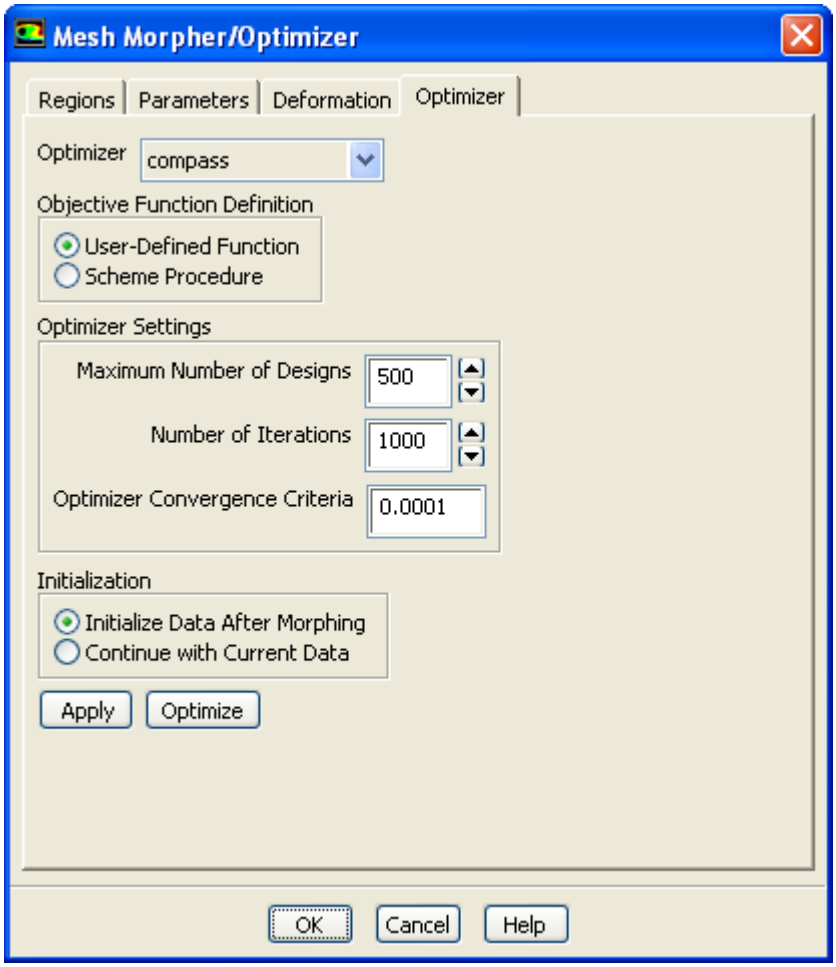
m. Repeat steps 6.a.-l. for each additional region in which you want to apply parameters to control points.

7. If you are using the built-in optimizers, define the optimizer settings.

   a. Click the **Optimizer** tab to reveal the settings shown in *Figure 12.5* (p. 641).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.5 The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box



b. Select an optimizer from the **Optimizer** drop-down list. The available optimizers are not based on gradients, and include the following: **compass**, **powell**, **rosenbrock**, **simplex**, and **torczon**. For more information about how these optimizers function, see *Optimizers* (p. 632).

c. Make a selection in the **Objective Function Definition** group box, to specify whether the objective function that will be minimized during the optimization process was provided in step 2. as a **User-Defined Function** or as a **Scheme Procedure**.

d. Define the settings in the **Optimizer Settings** group box.

   i. Enter a value for **Maximum Number of Designs**, to specify the maximum number of design stages the optimizer will undergo to reach the specified objective function. Note that this number is not the number displayed in the console during the optimization process. Each optimizer undergoes a certain number of design modifications (called "runs") before it converges to a design for a particular design stage. The Run number displayed in the console refers to the actual design modifications (including the inner loop) irrespective of the optimizer, whereas the **Maximum Number of Designs** value is the maximum number of converged designs provided by the optimizer.

   ii. Enter the **Number of Iterations**, to specify the maximum number of iterations ANSYS FLUENT performs for each design change.

   iii. Enter a number for **Optimizer Convergence Criteria**, to specify the convergence criteria for the optimizer.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Specify how the solution variables should be treated after the mesh is deformed by making a selection in the **Initialization** group box. Select **Initialize Data After Morphing** to specify that the solution variables should be initialized to the values specified in the **Solution Initialization** task page after deformation. Select **Continue with Current Data** to specify that the solution variables remain the values obtained in the previous design iteration.

f. Click the **Apply** button to save your optimizer settings.

g. Click the **Optimize** button to initiate the optimization process. Information about each run (i.e., each design modification) that is generated as part of the production of a design stage will be displayed in the console, and the deformed mesh for each run will be updated in the graphics window.

8. Click **OK** to close the **Mesh Morpher/Optimization** dialog box.

9. Run your case file with the deformed mesh.

10. If you want to rerun a case file in which the mesh has been deformed by the **Optimize** button in the **Optimizer** tab, you must revise the settings in the **Mesh Morpher/Optimizer** dialog box so that they are suitable for the revised mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 13: Modeling Turbulence

This chapter provides details about how to use the turbulence models available in ANSYS FLUENT.

Information about turbulence modeling theory is presented in "Turbulence" in the Theory Guide. Information about using the turbulence models can be found in the following sections:

13.1. Introduction
13.2. Choosing a Turbulence Model
13.3. Steps in Using a Turbulence Model
13.4. Setting Up the Spalart-Allmaras Model
13.5. Setting Up the k- ε Model
13.6. Setting Up the k- ω Model
13.7. Setting Up the Transition k- kl- ω Model
13.8. Setting Up the Transition SST Model
13.9. Setting Up the Reynolds Stress Model
13.10. Setting Up the Scale-Adaptive Simulation (SAS) Model
13.11. Setting Up the Detached Eddy Simulation Model
13.12. Setting Up the Large Eddy Simulation Model
13.13. Setting Up the Embedded Large Eddy Simulation (E-LES) Model
13.14. Setup Options for all Turbulence Modeling
13.15. Defining Turbulence Boundary Conditions
13.16. Providing an Initial Guess for k and ε (or k and ω)
13.17. Solution Strategies for Turbulent Flow Simulations
13.18. Postprocessing for Turbulent Flows

13.1. Introduction

Turbulence is the three-dimensional unsteady random motion observed in fluids at moderate to high Reynolds numbers. As technical flows are typically based on fluids of low viscosity, almost all technical flows are turbulent. Many quantities of technical interest depend on turbulence, such as:

- Mixing of momentum, energy and species
- Heat transfer
- Pressure losses and efficiency
- Forces on aerodynamic bodies
- etc.

While turbulence is, in principle, described by the Navier-Stokes equations, it is not feasible in most situations to resolve the wide range of scales in time and space by Direct Numerical Simulation (DNS) as the CPU requirements would by far exceed the available computing power for any foreseeable future. For this reason, averaging procedures have to be applied to the Navier-Stokes equations to filter out all, or at least, parts of the turbulent spectrum. The most widely applied averaging procedure is Reynolds-averaging (which, for all practical purposes is time-averaging) of the equations, resulting in the Reynolds-Averaged Navier-Stokes (RANS) equations. By this process, all turbulent structures are eliminated from the flow and a smooth variation of the averaged velocity and pressure fields can be obtained. However, the averaging process introduces additional unknown terms into the transport equations (Reynolds Stresses and Fluxes) which need to be

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

provided by suitable turbulence models (turbulence closure). The quality of the simulation can depend crucially on the selected turbulence model and it is important to make the proper model choice as well as to provide a suitable numerical grid for the selected model. An alternative to RANS are Scale-Resolving Simulation (SRS) models. With SRS methods, at least a portion of the turbulent spectrum is resolved in at least a part of the flow domain. The most well-known such method is Large Eddy Simulation (LES), but many new hybrids (models between RANS and LES) are appearing. As all SRS method require time-resolved simulations with relatively small time steps, it is important to understand that these methods are substantially more computationally expensive than RANS simulations.

ANSYS FLUENT provides the following choices of turbulence models:

- Spalart-Allmaras model
- k- ε models
  - Standard k- ε model
  - Renormalization-group (RNG) k- ε model
  - Realizable k- ε model
- k- ω models
  - Standard k- ω model
  - Shear-stress transport (SST) k- ω model
- v^2-f model (add-on)
- Transition k- kl- ω model
- Transition SST model
- Reynolds stress models (RSM)
  - Linear pressure-strain RSM model
  - Quadratic pressure-strain RSM model
  - Low-Re stress-omega RSM model
- Scale-Adaptive Simulation (SAS) model
- Detached eddy simulation (DES) model, which includes one of the following RANS models.
  - Spalart-Allmaras RANS model
  - Realizable k- ε RANS model
  - SST k- ω RANS model
- Large eddy simulation (LES) model, which includes one of the following sub-scale models.
  - Smagorinsky-Lilly subgrid-scale model
  - WALE subgrid-scale model
  - Dynamic Smagorinsky model
  - Kinetic-energy transport subgrid-scale model

13.2. Choosing a Turbulence Model

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics of

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for your application, you need to understand the capabilities and limitations of the various options.

The purpose of this section is to give an overview of issues related to the turbulence models provided in ANSYS FLUENT. The computational effort and cost in terms of CPU time and memory of the individual models is discussed. While it is impossible to state categorically which model is best for a specific application, general guidelines are presented to help you choose the appropriate turbulence model for the flow you want to model.

For additional information, please see the following sections:

13.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models
13.2.2. Scale-Resolving Simulation (SRS) Models
13.2.3. Grid Resolution SRS Models
13.2.4. Numerics Settings for SRS Models
13.2.5. Model Hierarchy

13.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models

RANS models (Reynolds (Ensemble) Averaging in the Theory Guide) offer the most economic approach for computing complex turbulent industrial flows. Typical examples of such models are the k-ε or the k-ω models in their different forms. These models simplify the problem to the solution of two additional transport equations and introduce an Eddy-Viscosity (turbulent viscosity) to compute the Reynolds Stresses. More complex RANS models are available which solve an individual equation for each of the six independent Reynolds Stresses directly (Reynolds Stress Models – RSM) plus a scale equation (ε-equation or ω-equation). RANS models are suitable for many engineering applications and typically provide the level of accuracy required. Since none of the models is universal, you have to decide which model is the most suitable for a given applications.

13.2.1.1. Spalart-Allmaras One-Equation Model

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed specifically for aeronautics and aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity in turbomachinery applications. Do not use the model as a general purpose model, as it is not well calibrated for free shear flows (large errors for e.g., jet flows). The current model implementation can be run as a low Reynolds number model on fine near wall grids ($y^+ > 3$) or on wall function meshes ($y^+ > 30$). Avoid using intermediate grids, as currently now the Enhanced Wall Treatment (EWT) for the buffer layer is available for this model.

13.2.1.2. k-ε Models

Two-equation models are historically the most widely used turbulence models in industrial CFD. They solve two transport equations and model the Reynolds Stresses based using the Eddy Viscosity approach. The standard k-ε model in ANSYS FLUENT falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [*43*] (p. 2211). Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations.

The draw-back of all k-ε models is their insensitivity to adverse pressure gradients and boundary layer separation. They typically predict a delayed and reduced separation relative to observations. This can result in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overly optimistic design evaluations for flows which separate from smooth surfaces (aerodynamic bodies, diffusers, etc.). The k-ε model is therefore not widely used in external aerodynamics.

In ANSYS FLUENT, the use of the Realizable k-ε model is recommended relative to other variants of the k-ε family. It is recommended to use the k-ε model in combination with the Enhanced Wall Treatment (EWT). Refer to Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide for details. For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, etc.), k-ε models are generally not recommended.

13.2.1.3. k-ω Models

The ω-equation offers several advantages relative to the ε-equation. The most prominent one is that the equation can be integrated without additional terms through the viscous sublayer. This makes the formulation of robust y+-insensitive Enhanced Wall Treatment (EWT) relatively straightforward. Refer to Enhanced Wall Treatment ω-Equation (EWT-ω) in the Theory Guide for details. Furthermore, k-ω models are typically better in predicting adverse pressure gradient boundary layer flows and separation. The downside of the standard ω-equation is a relatively strong sensitivity of the solution depending on the freestream values of k- and ω- outside the shear layer. The use of the standard k-ω model is, for this reason, not generally recommended in ANSYS FLUENT.

The SST k-ω model has been designed to avoid the freestream sensitivity of the standard k-ω model, by combining elements of the ω-equation and the ε-equation. In addition, the SST model has been calibrated to accurately compute flow separation from smooth surfaces. Within the k-ω model family, it is therefore recommended to use the SST model. The SST model is one of the most widely used models for aerodynamic flows. It is typically somewhat more accurate in predicting the details of the wall boundary layer characteristics than the Spalart-Allmaras model. The SST model (as all ω-equation based models) uses the enhanced wall treatment as default.

For the k-ω models, so-called low Reynolds number terms (low Re) have been proposed by Wilcox. These are available in ANSYS FLUENT as an option. It is important to point out that these terms are not required for integrating the equations through the viscous sublayer. Their main influence lies in mimicking laminar-turbulent transition processes. However, this functionality is not widely calibrated and for wall-boundary layer transition, the combination of the SST model with the Transition SST model (Transition SST Model in the Theory Guide) or the k-kl transition model (k- kl- ω Transition Model in the Theory Guide) is more reliable. The use of the low-Re terms is therefore not encouraged.

13.2.1.4. RSM Models

Reynolds Stress models (RSM) include several effects which are not easily handled by Eddy-Viscosity models. The most important effect is the stabilization of turbulence due to strong rotation and streamline curvature (as observed e.g., in cyclone flows). RSM on the other hand often demand a significant increase in computing time partly due to the additional equations but mostly due to reduced convergence. This additional effort is not always justified by increased accuracy. Their usage is therefore not generally recommended and should be restricted to flows for which their superiority has been established, especially flow with strong swirl and rotation. If wall boundary layers are important, the combination of RSM and the ω-equation is more accurate than the combination with the ε-equation.

13.2.1.5. Laminar-Turbulent Transition Models

Two models for transition prediction are available in ANSYS FLUENT, namely the SST-transition model and the k-kl Transition model. For many test cases, both models produce similar results. Due to its combination with the SST model, the SST-Transition model is favored. It is important to point out that only laminar-turbulent transition of wall boundary layers can be simulated with any of these two models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Proper mesh refinement and specification of inlet turbulence levels is crucial for accurate transition prediction. In general, there is some additional effort required during the mesh generation phase because a low-Re mesh with sufficient streamwise resolution is needed to accurately resolve the transition region. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary in order to properly capture the rapid transition due to the separation bubble. Finally, the decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location. Physically correct values for the turbulence intensity (Tu) should be achieved near the location of transition.

13.2.1.6. Model Enhancements

There are many model enhancements available for turbulence models. While such enhancements can improve simulations in some cases, they can also have detrimental effects. The general recommendation is therefore to use them with caution.

As noted above, the use of low-Re terms in combination with the ω-equation is not generally recommended.

Another model enhancement which should be used with care is the compressibility correction of Sarkar ([*73*] (p. 2212)). It can improve the prediction of free shear layers at high Mach numbers, but has also shown a pronounced negative impact on wall boundary layers. It is therefore not generally recommended. For historic reasons it is set to default for the compressible solver. It is advised to turn it off by hand unless the application is mainly to free shear flows.

Buoyancy has a pronounced effect on turbulence (Effects of Buoyancy on Turbulence in the Theory Guide). The use of the source term in the k-equation is therefore recommended for buoyancy affected flows. The source term in the ε-equation and ω-equation is much less established and the term should be activated with care.

13.2.1.7. Wall Treatment RANS Models

It is recommended to use the Enhanced Wall Treatment for all models for which it is available (ε-equation and ω-equation). It provides the most consistent wall shear stress and wall heat transfer predictions with the least sensitivity to y+ values. (see Enhanced Wall Treatment ε-Equation (EWT-ε) and Enhanced Wall Treatment ω-Equation (EWT-ω) in the Theory Guide for more information)

For the Spalart-Allmaras model use either a grid with $y^+ > 30$ or $y^+ < 3$ (recommended), as no Enhanced Wall Treatment is available for this model.

In case Wall Functions are used, it is necessary to avoid grids with fine near wall spacing. It is recommended to use $y^+ > 30$ in the entire domain. The application of Wall Functions is, however, not generally recommended as they do not allow a systematic refinement of the near wall grid. Wall Functions are especially damaging for flows at low to medium Reynolds numbers (Re~10^4-10^6), as the assumption of an extended logarithmic layer is not valid in these cases. In case that Wall Functions are desired, the option of Scalable Wall Functions (Scalable Wall Functions in the Theory Guide) avoids the grid restrictions and can be run on fine meshes.

13.2.1.8. Grid Resolution RANS Models

Grid generation has a strong impact on model accuracy. There are many considerations which have to be followed when generating high quality CFD grids. From a turbulence modelling standpoint, the most important one is that the relevant shear layers should be covered by at least ~10 cells normal to the layer. Below this resolution, the model will not be able to provide its calibrated performance. Especially for free shear flows,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

whose location is not known during grid generation, this is a requirement which is hard to achieve. Nevertheless, you should be aware that for lower resolution, the model performance can degrade.

For wall bounded flows, a structured mesh in wall-normal direction is highly recommended. The structured portion of the mesh should cover the entire boundary layer and extend beyond the boundary layer thickness to avoid restricting the growth of the boundary layer. Advanced turbulence models for wall boundary layers like the Spalart-Allmaras model and the SST model will only provide improved results to other models if a minimum of 10 or more structured (hex or prism) cells are located inside the boundary layer. In addition, one should ensure that the prism layer covers the wall boundary layer entirely. Note that these are not specific requirements of these models, but are general requirements for wall boundary layer simulations.

Both, ε-based and ω-based models offer Enhanced Wall Treatment options (see Enhanced Wall Treatment ε-Equation (EWT-ε) and Enhanced Wall Treatment ω-Equation (EWT-ω) in the Theory Guide for more information), which make the models relatively insensitive to the y^+-value of the wall cell. Generally speaking, it is more important to ensure that the boundary layer is covered with sufficient cells, then to achieve a certain y^+ criterion. However, for simulations with high accuracy demands on the wall boundary layer (especially for heat transfer predictions) near wall meshes with y^+~1 are recommended. When wall functions are used, it is essential to avoid meshes with y^+ values lower than ~30 as the wall shear stress and the wall heat transfer can and will seriously deteriorate under such conditions. For this reason, the usage of Enhanced Wall Treatments (to be selected for ε-equation and default for ω-equation based models) is recommended.

For transition models (see k- kl- ω Transition Model and Transition SST Model in the Theory Guide), more stringent grid resolution requirements apply than for standard RANS models, as transition modelling requires the resolution of the thin laminar boundary layer upstream of the transition location. For this reason, a low-Re mesh ($y^+ < 1$) with sufficient streamwise resolution is needed to accurately resolve the transition region. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary, in order to properly capture the rapid transition due to the separation bubble. Finally, the decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location.

13.2.2. Scale-Resolving Simulation (SRS) Models

The alternative to RANS models are models which resolve at least a portion of the turbulence for at least a portion of the flow domain. Such models are generally termed 'Scale-Resolving'.

13.2.2.1. Large Eddy Simulation (LES)

The most widely know such modelling concept is Large Eddy Simulation (LES). It is based on the approach of resolving large turbulent structures in space and time down to the grid limit everywhere in the flow. However, while widely used in the academic community, LES had very limited impact on industrial simulations. The reason lies in the excessively high resolution requirements for wall boundary layers. Near the wall, the largest scales in the turbulent spectrum are nevertheless geometrically very small and require a very find grid and a small time step. In addition, unlike RANS, the grid cannot only be refined in the wall normal direction, but also needs to resolve turbulence in the wall parallel plane. This can only be achieved for flows at very low Reynolds number and on very small geometric scales (the extent of the LES domain cannot be much larger than 10-100 times the boundary layer thickness parallel to the wall). For this reason the use of LES is only recommended for flows where wall boundary layers are not relevant and need not be resolved or for flows where the boundary layers are laminar due to the low Reynolds number. In such cases, the most balanced LES model is the WALE model (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide). It offers a good compromise between model complexity and generality. It also allows computing laminar shear (boundary) layers without any model impact.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.2.2.2. Hybrid RANS-LES Models

In order to avoid the high resolution requirements of LES, numerous hybrid models have been developed in recent years. These are models which combine certain elements of RANS and LES approaches in a way which allows the simulation of high Reynolds number flows. With hybrid models, the attached wall boundary layers are typically covered by the RANS part of the model, while large detached regions are handled in 'LES' mode, meaning with a partial resolution of the turbulent spectrum in space and time. Hybrid models rely on a strong enough flow instability to generate turbulent structures in the separated zone. This is typically the case for flows behind bluff bodies, where URANS models predict single-mode periodic vortex shedding. Hybrid models allow these vortices to generate smaller eddies down to the available grid limit. *Figure 13.1* (p. 649) shows a typical scenario: While the application of a standard RANS model in unsteady mode results in a single frequency vortex shedding (left), the application of hybrid models allows a break-up of the large structures into smaller scales. This is beneficial for predicting the correct mixing behind the body or to extract spectral information (e.g., for acoustic simulations). At the same time, the wall boundary layers are covered by the RANS part of the hybrid model avoiding the excessive resolution requirements of LES.

Figure 13.1 Illustration of SST-URANS vs. SST-SAS Models

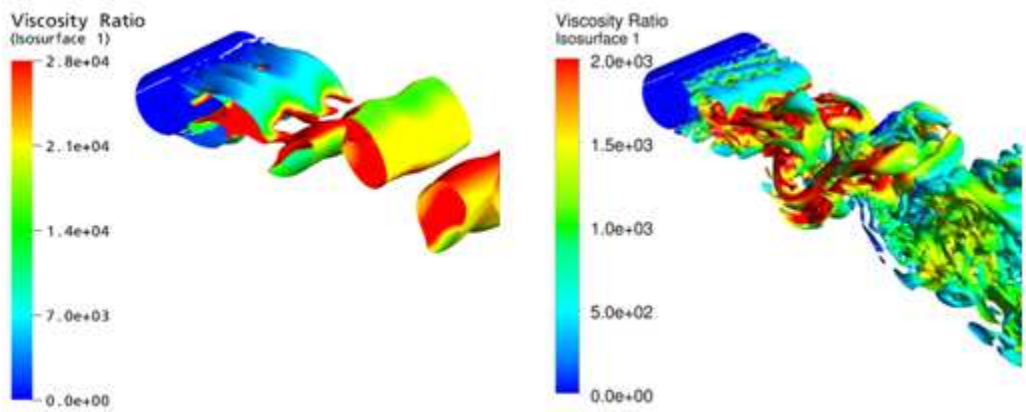


Figure 13.1 (p. 649) shows a circular cylinder in a cross flow at $\mathrm{Re}=3.6\times10^{6}$. The left hand side illustrates the SST-URANS model, while the right-hand side illustrates the SST-SAS model. The iso-surface of $\mathrm{Q}=\mathrm{S}^{2}-\Omega^{2}$ is colored according to the eddy viscosity ratio μ_t/μ (note that the scale in the right-hand side figure is smaller by a factor of 14).

13.2.2.2.1. Scale-Adaptive Simulation (SAS)

The SAS modelling approach (see Scale-Adaptive Simulation (SAS) Model in the Theory Guide) as proposed by Menter et al ([*53*] (p. 2211)[*54*] (p. 2211)) is based on the introduction of the von Karman length scale, L_{vK}, into the turbulence equations (in case of the SST model it enters into the ω-equation). L_{vK} is defined as the ratio of the first divided by the second derivative of the velocity vector (times the von Karman constant κ=0.41):

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$L_{vK}=\kappa\left|\frac{U'}{U''}\right|;\quad U''=\sqrt{\frac{\partial^2 U_i}{\partial x_k^2}\frac{\partial^2 U_i}{\partial x_j^2}};\quad U'=S=\sqrt{2\cdot S_{ij}S_{ij}};\quad S_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) \tag{13–1}$$

The inclusion of this term allows the model to adjust its length scale to already resolved scales in the flow and thereby provide a low enough eddy viscosity to allow the model to operate in 'LES' mode.

The SAS approach has the advantage that the RANS part of the model is unaffected by the grid spacing and will therefore not allow a deterioration of model accuracy as seen in DES in regions of refined grid but insufficient flow instability. However, in cases where the flow instability is not strong enough, the SAS will remain in RANS mode and will not produce unsteady structures. While this is often a sign that the RANS model is still reasonably capable of handling the flow, it is a limitation if unsteady information is required (e.g., in acoustics). In such cases, the internal interface option of the E-LES implementation can be applied to convert modeled turbulence into resolved turbulence (see Internal Interface Without LES Zone in the Theory Guide).

13.2.2.3. Detached Eddy Simulation (DES)

The DES model (see Detached Eddy Simulation (DES) in the Theory Guide) achieves the switch between RANS and LES by a comparison of the turbulent length scale L_t with the grid spacing (maximum of cell edge). The model selects the minimum of both and thereby switches between RANS and LES mode by replacing ε in the k-equation by:

$$\varepsilon=\frac{k^{3/2}}{L_t}\quad\rightarrow\quad\varepsilon=\frac{k^{3/2}}{\min(L_t,c_{DES}\Delta_{\max})} \tag{13–2}$$

Once the model selects the grid spacing as the minimum, the model is operating in 'LES' mode.

The grid spacing enters explicitly into the DES model. This can affect the RANS solution in regions, where the grid is between RANS and LES resolution (so-called 'grey zones' in DES) and/or where the flow instability is not strong enough to generate LES structures. Another issue to consider with DES is the problem of 'grid-induced-separation' (GIS). It occurs if the grid for an attached wall boundary layer flow is refined to a point where the DES limiter becomes active and affects the RANS solution. However, in such situations, the flow instability is not strong enough to balance the reduced RANS content by resolved turbulence. This will typically result in an artificial flow separation at the location of grid refinement. It typically happens if $\Delta_{\max}<\delta$ (δ being the boundary layer thickness). Remedies for this situation have been proposed by Menter et al. who recommended using the F1 and F2 blending functions of the SST-DES model to shield the boundary layers from the DES limiter. Later, alternative blending functions for the same purpose have been proposed by Spalart et al. ([*74*] (p. 2213)) – resulting in the terminology Delayed DES (DDES). Note however, that the DDES model only offers limited protection against GIS and the usage of the F2-SST-blending function is recommended. For the k-ε and the Spalart-Allmaras model only the DDES shielding functions are available and should be used with care.

A further refinement is provided by the Improved DDES (IDDES) formulation of Strelets et al. ([*79*] (p. 2213)), which extends the LES zone of the model to the outer part of wall boundary layers. This allows the simulation of wall boundary layers in Wall Modeled LES (WMLES) mode. In this model, the IDDES model is applied like a LES model, typically with the specification of unsteady inlet conditions. The grid resolution requirements for WMLES are much less stringent than for LES.

All the above variants are available in ANSYS FLUENT. Currently the usage of the F2-SST blending function is recommended and set as default in the SST-DDES formulation. In the k-ε and the Spalart-Allmaras model, the standard DDES functions are default. When using these DDES formulations, you should be aware that

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the shielding function offers only limited protection against GIS. Despite the potential difficulties in the application of hybrid methods, they have the potential to greatly expand the usage of Scale-Resolving Simulation models for engineering applications, as they avoid the excessive resolution required by LES for wall boundary layers. IDDES in WMLES mode is an advanced option and should only be used if you are familiar with the original literature and the grid requirements for this model.

13.2.2.4. Zonal Modeling and Embedded LES (E-LES)

As pointed out in the previous sections, hybrid models rely on flow instabilities to generate turbulent structures in large separated regions without the explicit introduction of unsteadiness through the boundary conditions. However, there are situations, where such instabilities are not present or are not reliable to serve this purpose. In such cases, it is desirable to apply RANS and the LES models in predefined zones and provide clearly defined interfaces between them. At these interfaces, the modeled turbulent kinetic energy from the upstream RANS model is converted explicitly to resolved scales at an internal boundary to the LES zone. The LES zone can then be limited to the region of interest where unsteady results are required. E-LES is available in ANSYS FLUENT and allows the combination of most RANS model with classical LES models. It is important to emphasize that in this mode, a full LES resolution is required within the LES zone. In the LES zone, using the WALE model is recommended (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide).

13.2.3. Grid Resolution SRS Models

13.2.3.1. Wall Boundary Layers

For a wall-resolved LES, it is typically recommended to use a mesh with a grid spacing scaling with $\Delta x^+ \approx 40$, $\Delta y^+ \approx 20$, $\Delta z^+ \approx 20$ where x is the streamwise, y the wall normal and z the spanwise direction (e.g., channel flow). However, in complex applications, the distinction between streamwise and spanwise direction is not feasible and then $\Delta x^+ \approx 20$, $\Delta y^+ \approx 20$, $\Delta z^+ \approx 20$ would be required. This scaling demonstrates the strong Reynods number dependency of the LES approach for wall bounded flows.

For the IDDES model in WMLES mode, the above requirements can be relaxed. The grid spacing no longer scale with the wall friction, but with the boundary layer thickness δ. It is recommended to use $\Delta x \approx \Delta z \approx (0.05 - 0.1) \cdot \delta$. The wall normal resolution should be like for a finely resolved RANS simulation, meaning a near wall resolution of $\Delta y^+ \approx 1$ and around 30 nodes inside the boundary layer.

The other hybrid models, SAS and DDES, require near wall resolution as in the underlying RANS models, as the boundary layers are covered in RANS mode. For the DDES (and the IDDES model run in DDES mode) model, Grid Induced Separation (GIS) may occur when not using the SST-F2 shielding function for $\Delta x_{\max} < \delta$. For the SST-F2 function the value is more $\Delta x_{\max} < 0.1 \cdot \delta$. This does however to a certain extent depend on the pressure gradient. Note however, that the SST-F2 function can reduce the effectiveness of the DDES model to switch into scale-resolving mode.

13.2.3.2. Free Shear Flows

For free shear flows, it is difficult to provide general recommendations, as there are many different flow scenarios. The current recommendation is therefore based on the most common (and most frequent) free shear flow – a turbulent mixing layer. It will not necessarily apply to other free shear flows like jets and you are advised to perform tests as to the optimal resolution of your specific flow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For free shear flows and SRS models, one should aim for uniform isotropic cells (all edges have similar length). The shear layer should be covered by ~10-20 cells. As the layer thickness is often hard to estimate, one can also look at the ratio

$$R_l = \frac{L_t}{\Delta_{\max}}; \quad L_t = \frac{k^{3/2}}{\varepsilon} = \frac{k^{1/2}}{0.09\omega} \tag{13–3}$$

Where $\Delta\max$ is the maximum cell edge length. It is available in ANSYS CFD-Post for all turbulence models. This estimate should be based on the RANS solution and not on the SAS/DES solution for the given flow! The above requirement for the mixing layer translates into:

$$R_l > 5 - 10 \tag{13–4}$$

In these estimates, 10 cells across the layer (or $R_l = 5$) is truly a lower limit. The higher values of 20 cells and $R_l = 10$ are much safer.

13.2.4. Numerics Settings for SRS Models

For Scale-Resolving Simulations (SRS), specific discretization and solver settings are required to achieve optimal accuracy with minimal numerical effort. The recommendations given below should be considered as a starting point for your specific flow application and are not generic, but problem dependent. The recommendations are based on incompressible, single phase flow without chemical reactions or other complex additional physics. In case your simulation features additional complex physical effects, it requires adjusting the recommended solver settings accordingly. In most cases, this will mean that a higher effort needs to be invested into the coupling of the equations (e.g. lower time step, reduced under-relaxation, higher iteration count, smaller residuals), in order to avoid a de-coupling of different physical phenomena.

For SRS models, it is generally recommended to use the pressure-based solver, as it offers optimized schemes for resolving turbulence relative to the density based solver.

For Scale-Resolving Simulations, optimal numerics settings are essential for achieving accurate results in an acceptable time frame. The reason is that at the SRS models operate at the resolution limit of the provided grid where the smallest scales are of the order of the grid spacing and the time resolution. Numerics settings therefore have to be chosen to provide an optimal balance between accuracy and robustness.

It is generally recommended to initialize the solution from a (reasonably) converged RANS simulation.

13.2.4.1. Time Discretization

For Scale-Resolving Simulations, the resolution of the turbulent structures in time is essential for the success of the simulation. This is, to the largest extent, defined by the selected time step. As the SRS model is operating at the grid limit, you should select a time step that ensures a Courant-Friedrichs-Levy (CFL) number of

$$CFL = \frac{U\Delta t}{\Delta x} < 1 \tag{13–5}$$

The CFL number is computed by the solver and can be checked based on an initial RANS simulation, for example. It is important to emphasize that this is not a numerical limit and that the solver will be able to sustain much larger CFL numbers. In complex applications, there will always be limited regions of fine cells

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or large velocities and you should not restrict the CFL number based on such zones. The recommendation of CFL =1 should be applied in the main SRS region in combination with a uniform isotropic grid. It is recommended to vary the time step for each type of application and explore its optimal value. This can substantially save on computing costs.

The time derivative should be computed by the Second Order Implicit option. (See *User Inputs for Time-Dependent Problems* (p. 1256) for guidelines on setting solution parameters for transient calculations in general.)

13.2.4.2. Spatial Discretization

For SRS models, it is important to minimize the numerical dissipation of the scheme in order to avoid damping of the smallest scales by numerical dissipation. For spatial discretization, the choice is between the Central Difference (CD) scheme (see Central-Differencing Scheme in the Theory Guide) and the Bounded Central Difference scheme (BCD) (see Bounded Central Differencing Scheme in the Theory Guide). The Central Difference scheme is the least dissipative and provides the highest resolution accuracy for the smallest scales. Especially for aero-acoustics simulations, where the spectral content at higher frequencies can be important, this is a desirable feature. However, Central Difference schemes are prone to solution oscillations (checker boarding) in the velocity field. When using the Central Difference scheme, it is therefore important to provide a high quality mesh (no mesh jumps, isotropic cells and high resolution in critical zones) and to avoid large time steps (the CFL number should be smaller than 1 in the main SRS region). It is recommended to visually monitor the solution regularly in order to avoid wasting computational resources. In case oscillations appear, the choice is to: improve the mesh; reduce the time step; or to switch to the slightly more dissipative, but also more robust Bounded Central Difference scheme. In many complex applications, the Bounded Central Difference Scheme is the numerics option of choice. It typically provides sufficiently low dissipation to allow the turbulent structures to evolve, but, at the same time, is robust enough to handle non-optimal grids as they are typically encountered in industrial simulations. In addition, the Bounded Central Difference scheme is also suitable for hybrid methods like SAS and DES, and will provide stable solutions in RANS regions, with highly stretched grids and with a CFL number larger than 1. For E-LES, the numerical scheme in the LES zone can be selected independently from the settings in the RANS zone. The RANS region can then be computed with standard higher order upwind schemes and the LES zone can be covered by either the Central Difference scheme or the Bounded Central Difference scheme.

For gradient discretization, it is recommended to select the Least Squares Cell Based option, or the Green-Gauss Node Based option, to ensure a second order interpolation on non-orthogonal grids.

For pressure interpolation, it is recommended to use the second order scheme, or the body-force-weighted scheme. Due to its higher dissipation, the PRESTO! scheme (see Pressure Interpolation Schemes in the Theory Guide) can result in a delayed formation or damping of turbulent structures. It is therefore not recommended for SRS.

13.2.4.3. Iterative Scheme

The selection of the iterative scheme will mostly affect the computational costs as the cost per iteration between these methods is rather different. However, recommendations are not straightforward, as the higher cost per iteration of a scheme can be offset by faster convergence within the time step.

The fastest scheme is the Non-Iteration Time Advancement (NITA) scheme (see Non-Iterative Time-Advancement Scheme in the Theory Guide as well as *Setting Solution Controls for the Non-Iterative Solver* (p. 1219)). This scheme typically works well for limited LES zones and high quality meshes. It is also important to ensure a low time step of a CFL number below 1. For the NITA scheme, all explicit under-relaxation factors are, by default, set equal to one. With NITA, it has slight advantages to use the fractional step method such that there is no involvement of more complex physics when the PISO scheme can be more beneficial.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In case the application is too complex for the NITA scheme to provide a solution, the iterative SIMPLEC (*SIMPLE vs. SIMPLEC* (p. 1216)) or PISO (*PISO* (p. 1216)) schemes are the next possible option. They should be preferred relative to the SIMPLE scheme, as they show faster convergence per time step and can be run with more aggressive under-relaxation (higher values equal to 1 or close to 1). If such settings prevent convergence within the time step, then check if your time step is small enough for maintaining CFL numbers below 1 in the SRS region – if not, then try reducing Δt. If this is not possible, or does not lead to satisfactory convergence, then reduce the under-relaxation factors to values between the default settings and 1. For skewed meshes or meshes with problematic quality, the reduction of explicit relaxation factor for pressure correction down to 0.7 from 1 can be very helpful.

13.2.4.3.1. Convergence Control

The convergence criterion within each time step will strongly affect solution costs, as a low criterion will result in an increased number of iterations. It is not possible to provide general recommendations, as the required residual depends on the application. The most relevant residual in the SRS models is the continuity residual. It should be converged by approximately 2 orders of magnitude per time step for a CFL number ~ 1. This should be achievable with around 5-10 maximum iterations per time step for flows involving no other physical models. Be sure to check the impact of convergence on your solution to ensure that the residual is reduced to a level consistent with your problem.

For simulations with small CFL values, the **Extrapolate variables** option (available in the *Run Calculation Task Page* (p. 1955)) can be very beneficial. This can substantially reduce the number of iterations within the time step up to 40% when the convergence control is based on the convergence criteria.

In case your simulation requires the combination of additional physical models, such as combustion or multiphase, a reduction of the under-relaxation factors might be required. In this case, the number of maximum iterations per time step can be increased to achieve the desired residual reduction.

For hybrid RANS/LES simulations, such as the SAS and DES models, there can be situations where the RANS portion of the flow limits convergence (e.g., due to poor grid quality). In such cases, the application of the coupled solver should be considered. For the coupled pressure-based solver, each iteration is typically more expensive than for the SIMPLEC algorithm, but this is at least partly off-set by faster convergence. The recommendations concerning residuals are similar to the SIMPLEC method, but the maximal number of iterations can be reduced to values as low as 2-5 for highly unstable flows, and 5-10 for more sensible flows and acoustics simulations.

13.2.5. Model Hierarchy

As discussed, turbulence modelling is a balance between accuracy and cost. The recommendation is to use RANS models as much as possible and as long as they provide the accuracy required for the simulation. RANS models will remain the work-horse of turbulence modelling for many years to come. Within the RANS family, eddy-viscosity models are typically sufficient for most engineering flow simulations. The application of RSM is only recommended for flows which are known to systematically benefit from their usage and justify the increase in computing power.

In cases where steady RANS or URANS models cannot provide the accuracy or unsteady information required, it is recommended to switch to the SAS approach. It is relatively forgiving in terms of the grid resolution and will not deteriorate the results in case of insufficient resolution in the unsteady zone. The SAS model will only provide scale-resolution if a strong flow instability is present. Visual inspection (using iso-surfaces of the Q-criterion) will quickly allow a judgment if the model provides sufficient unsteadiness and resolution relative to the grid spacing. In such cases, the internal interface option of the E-LES implementation can be applied to convert modeled turbulence into resolved turbulence (see Internal Interface Without LES Zone in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DDES (DES is not recommended) models can allow the formation of unsteadiness even for cases where SAS remains stable. DDES does require a more carefully crafted grid in the LES zone due to the DES grid influence on the RANS solution. For the SST-DES model, use the default F2 blending function for shielding.

E-LES is recommended in cases where limited zones with high accuracy requirements are embedded inside a larger RANS zone. At the interface between the RANS and the LES zone, unsteady turbulence is generated either by the Spectral Synthesizer or the Vortex Method. For wall bounded flows, this method requires a very fine near wall resolution in the LES zone.

Pure LES should only be applied to free shear flows (e.g., combustion chambers without wall influence, etc.) or to very limited domains using meshes with fine LES near wall resolution. It is important to be aware of the strong increase of grid resolution requirements with Reynolds number for wall bounded flows. For wall boundary layers at higher Reynolds numbers, the IDDES model can be run in WMLES mode – meaning with specified unsteady inlet conditions.

13.3. Steps in Using a Turbulence Model

When your ANSYS FLUENT model includes turbulence you need to activate the relevant model and options, and supply turbulent boundary conditions. These inputs are described in this section.

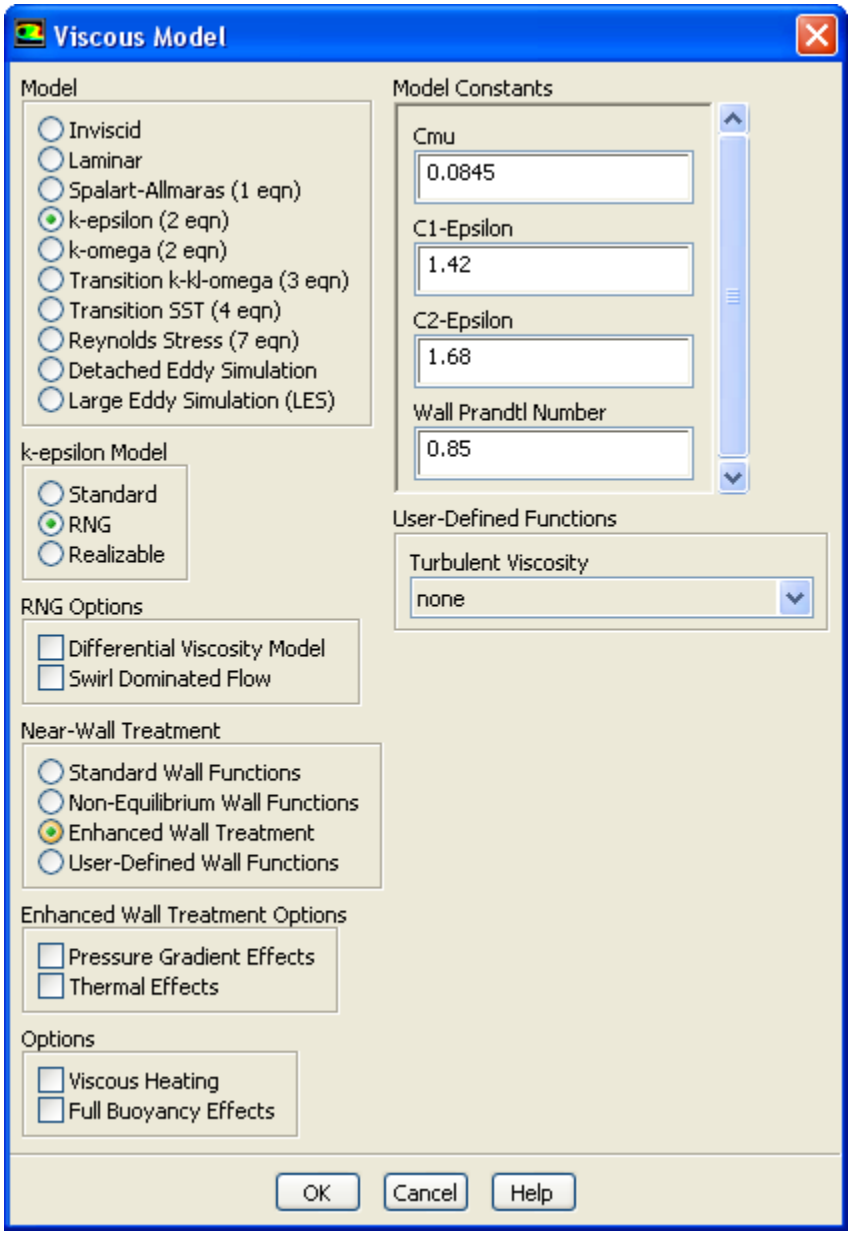
The procedure for setting up a turbulent flow problem is described below. (Note that this procedure includes only those steps necessary for the turbulence model itself; you will need to set up other models, boundary conditions, etc. as usual.)

1. To activate one of the turbulence models, select either **Spalart-Allmaras**, **k-epsilon**, **k-omega**, **Transition k-kl-omega**, **Transition SST**, **Reynolds Stress**, **Scale-Adaptive Simulation**, **Detached Eddy Simulation**, or **Large Eddy Simulation (LES)** under **Model** in the *Viscous Model Dialog Box* (p. 1645) (*Figure 13.2* (p. 656)).

   Models → **Viscous** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.2 The Viscous Model Dialog Box



If you choose the **k-epsilon** model, select either **Standard**, **RNG**, or **Realizable** under the **k-epsilon Model**. If you choose the **k-omega** model, select **Standard** or **SST** under **k-omega Model**.

Important

The **Detached Eddy Simulation** and the **Large Eddy Simulation (LES)** models are available only for 3D cases.

2. If the flow involves walls, and you are using one of the k- ε models or the RSM, choose one of the following options for the **Near-Wall Treatment** in the **Viscous Model** dialog box:
   - **Standard Wall Functions**
   - **Non-Equilibrium Wall Functions**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Enhanced Wall Treatment**
- **User-Defined Wall Functions**

For more information about these near-wall options, see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide. By default, the standard wall function is enabled.

For more information about the automatically defined near-wall treatment for the Spalart-Allmaras model, see Wall Boundary Conditions in the Theory Guide.

For more information about the automatically defined near-wall treatment for the k- ω model, see Wall Boundary Conditions in the Theory Guide.

For more information about the automatically defined near-wall treatment for the LES model, see Inlet Boundary Conditions for the LES Model in the Theory Guide.

3. Enable the appropriate turbulence modeling options in the **Viscous Model** dialog box. See *Setup Options for all Turbulence Modeling* (p. 673)
4. Specify the boundary conditions for the solution variables.

   Boundary Conditions

   See *Defining Turbulence Boundary Conditions* (p. 678) for details.
5. Specify the initial guess for the solution variables.

   Solution Initialization

   See *Providing an Initial Guess for k and ε (or k and ω)* (p. 681) for details. Note that Reynolds stresses are automatically initialized using k, and therefore need not be initialized explicitly.

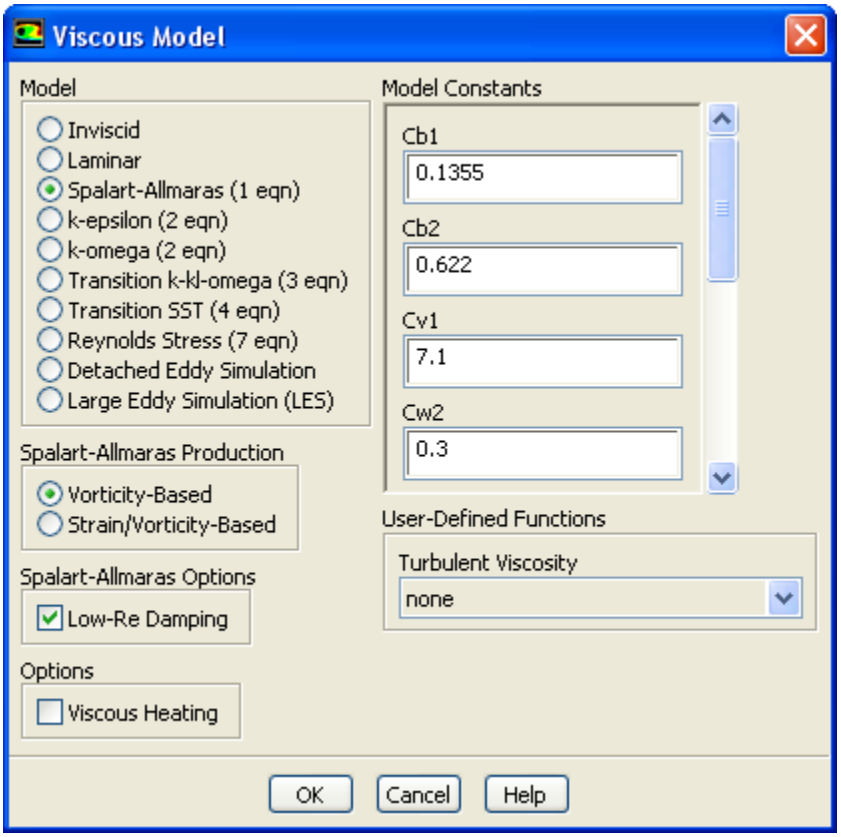
13.4. Setting Up the Spalart-Allmaras Model

If you choose the Spalart-Allmaras model, the following options are available:

- vorticity-based production (*Vorticity- and Strain/Vorticity-Based Production* (p. 674))
- strain/vorticity-based production (*Vorticity- and Strain/Vorticity-Based Production* (p. 674))
- low-Re damping (*Low-Re Damping* (p. 675))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.3 The Viscous Model Dialog Box Displaying the Spalart-Allmaras Production



13.5. Setting Up the k- ε Model

For additional information, please see the following sections:

13.5.1. Setting Up the Standard or Realizable k- ε Model
13.5.2. Setting Up the RNG k- ε Model

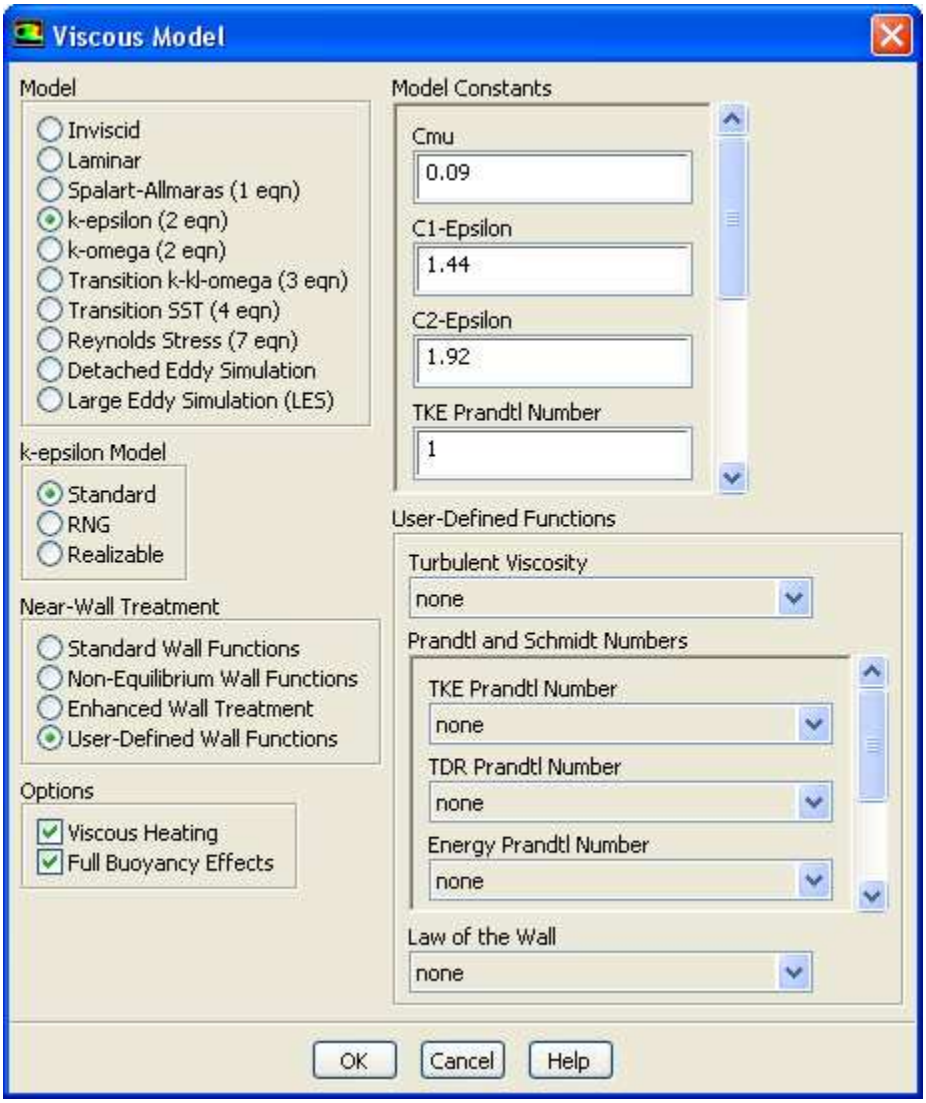
13.5.1. Setting Up the Standard or Realizable k- ε Model

If you choose the standard k- ε model or the realizable k- ε model, the following options are available:

- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k- ε Models in the Theory Guide)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.4 The Viscous Model Dialog Box Displaying the Standard k- ε Model



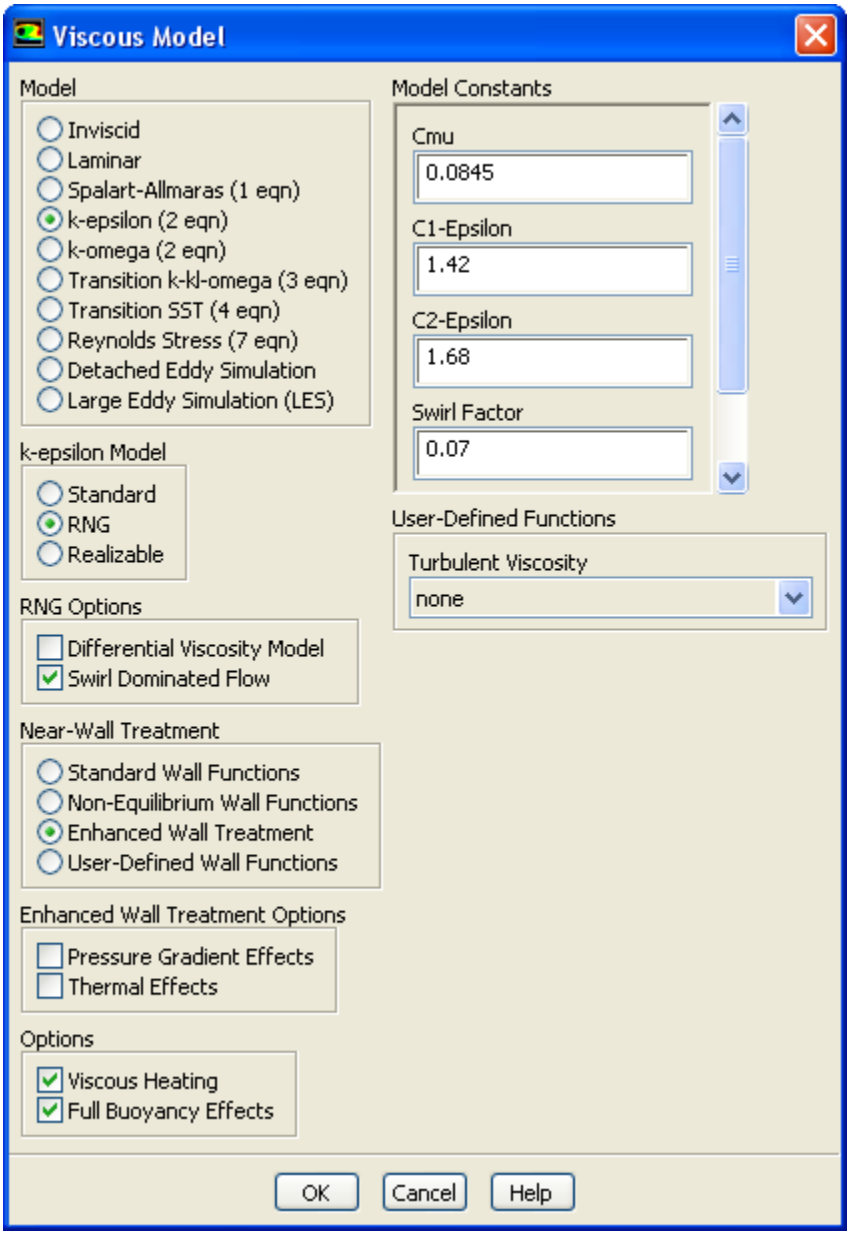
13.5.2. Setting Up the RNG k- ε Model

If you choose the RNG k- ε model, the following options are available:

- differential viscosity model (*Differential Viscosity Modification* (p. 674))
- swirl modification (*Swirl Modification* (p. 674))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k- ε Models in the Theory Guide)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.5 The Viscous Model Dialog Box Displaying the RNG k- ε Model



For all k- ε models, one the following near-wall treatments must be selected (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

- standard wall functions
- non-equilibrium wall functions
- enhanced wall treatment
- user-defined wall functions

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (*Including Pressure Gradient Effects* (p. 675))
- thermal effects (*Including Thermal Effects* (p. 676))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose the user-defined wall functions near-wall treatment, hook your UDF under **Law of the Wall**, as shown in *Figure 13.4* (p. 659).

13.6. Setting Up the k- ω Model

For additional information, please see the following sections:

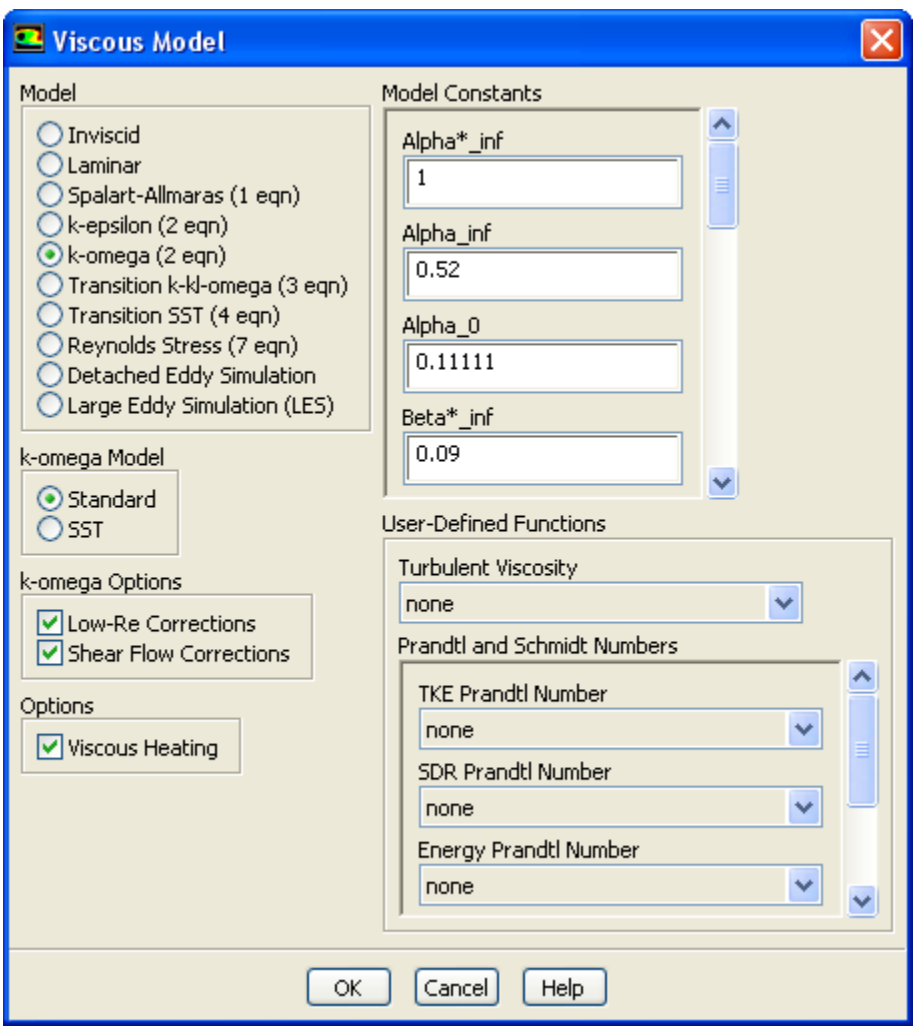
13.6.1. Setting Up the Standard k- ω Model
13.6.2. Setting Up the Shear-Stress Transport k- ω Model

13.6.1. Setting Up the Standard k- ω Model

If you choose the standard k- ω model, the following options are available:

- low-Re corrections (*Low-Re Corrections* (p. 675))
- shear flow corrections (*Shear Flow Corrections* (p. 675))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the immiscible fluid model) (*Turbulence Damping* (p. 675))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))

Figure 13.6 The Viscous Model Dialog Box Displaying the Standard k- ω Model



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

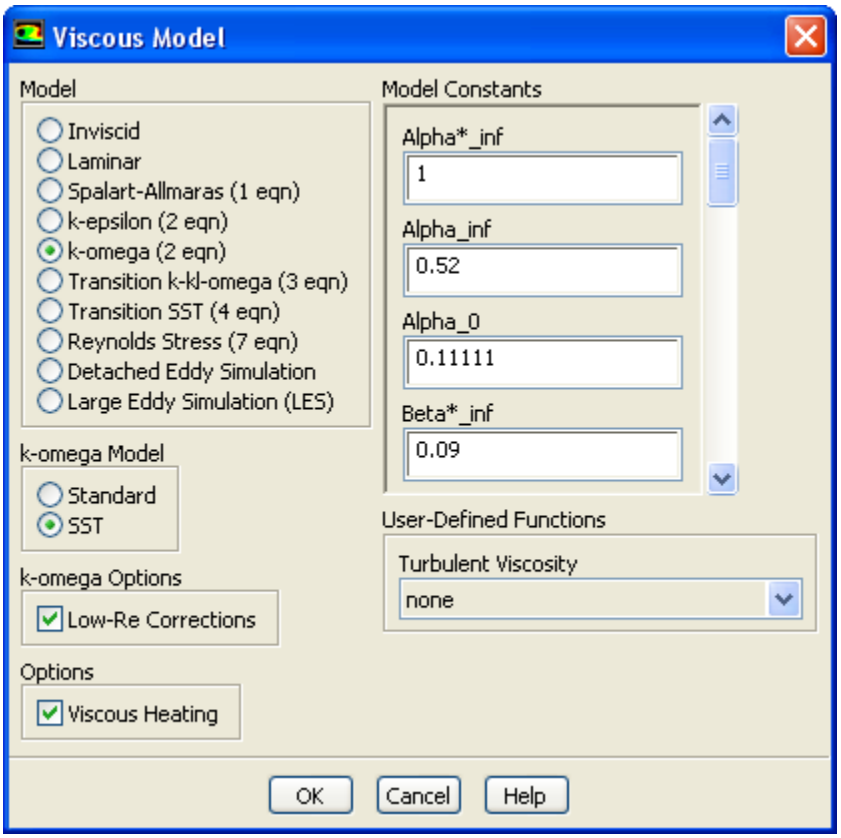
The k- ω models use enhanced wall functions as the near-wall treatment (see Enhanced Wall Treatment for Momentum and Energy Equations in the Theory Guide).

13.6.2. Setting Up the Shear-Stress Transport k- ω Model

If you choose the shear-stress transport k- ω model, the following options are available:

- low-Re corrections (*Low-Re Corrections* (p. 675))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the immiscible fluid model) (*Turbulence Damping* (p. 675))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))

Figure 13.7 The Viscous Model Dialog Box Displaying the SST k- ω Model



13.7. Setting Up the Transition k- kl- ω Model

If you choose the Transition k- kl- ω model, it is not necessary to modify any of the model constants.

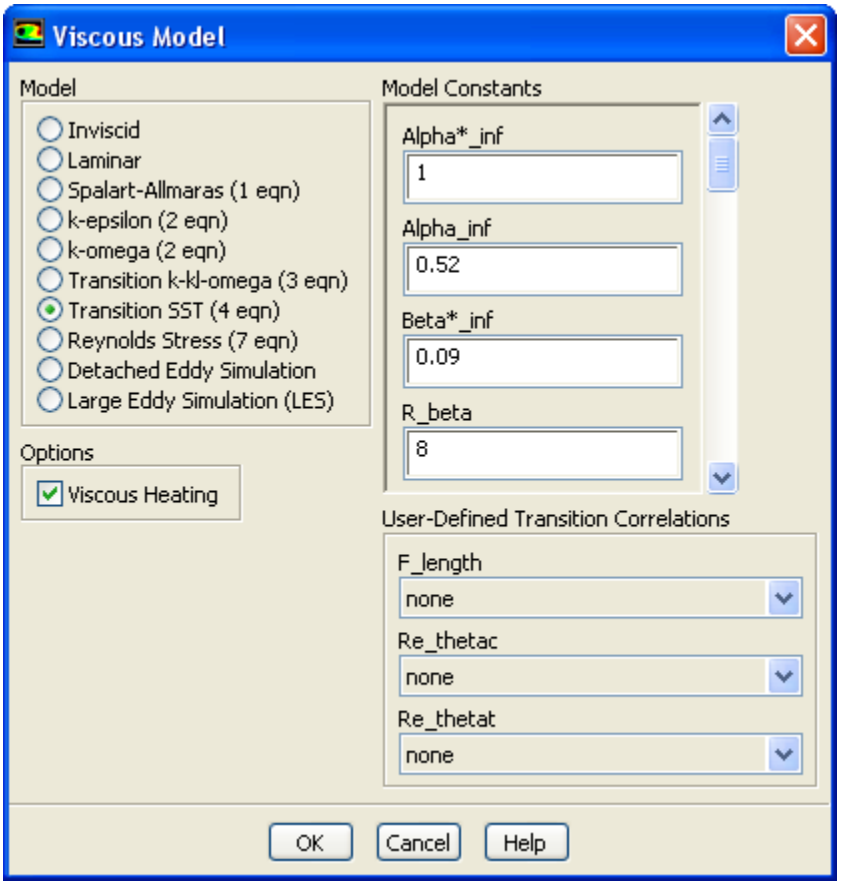
13.8. Setting Up the Transition SST Model

If you choose the Transition SST model, the following option is available:

- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.8 The Viscous Model Dialog Box for the Transition SST Model



You can customize your transition correlations, which are used in conjunction with the Transition SST model. The user-defined functions that you can hook are:

- transition length function (**F_length**).
- critical momentum thickness Reynolds number (**Re_thetac**).
- transition onset momentum thickness Reynolds number (**Re_thetat**).

For detailed information about the transition correlation UDFs, see DEFINE_TRANS UDFs in the UDF Manual.

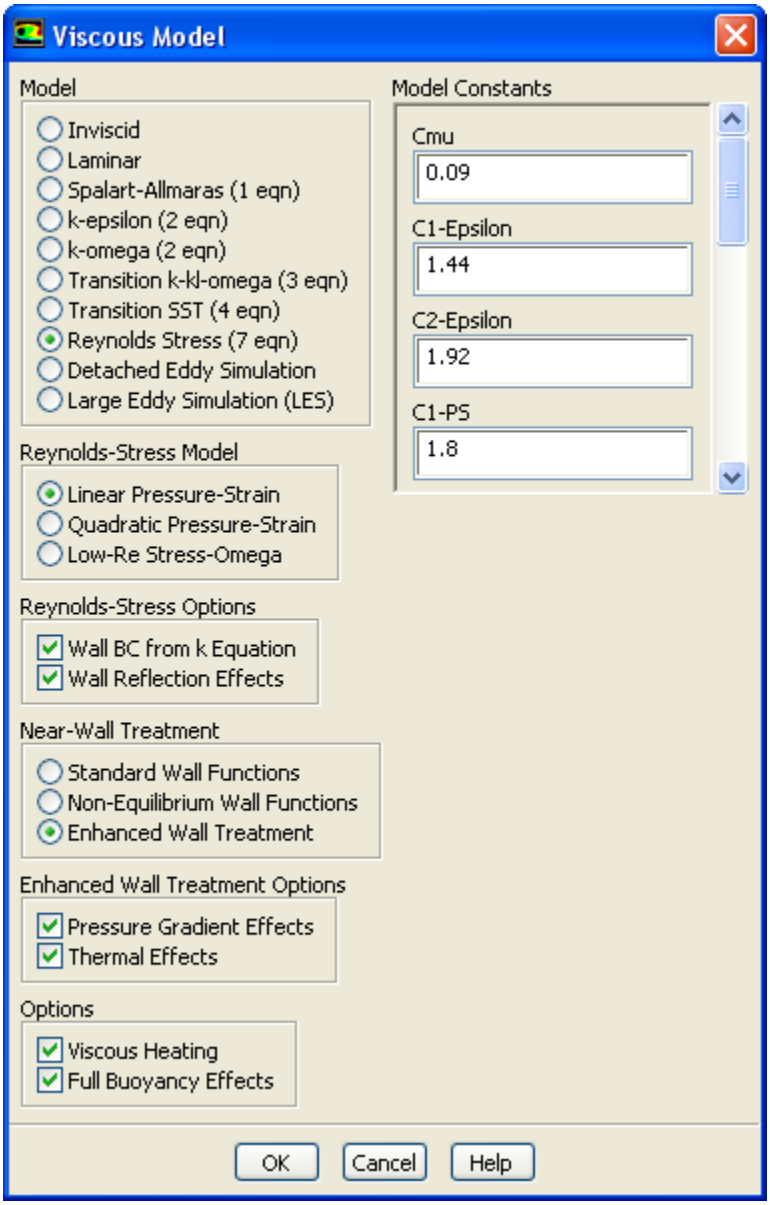
13.9. Setting Up the Reynolds Stress Model

If you choose the RSM, the following submodels are available:

- linear pressure-strain model (see Linear Pressure-Strain Model in the Theory Guide)
- quadratic pressure-strain model (*Quadratic Pressure-Strain Model* (p. 676))
- low-Re Stress-Omega (*Low-Re Stress-Omega Pressure-Strain* (p. 677))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.9 The Viscous Model Dialog Box Displaying the Reynolds Stress Model Options



The following Reynolds-stress options are available:

- wall boundary conditions for the Reynolds stresses from the k equation (*Solving the k Equation to Obtain Wall Boundary Conditions* (p. 676)) for the linear and quadratic pressure-strain models
- wall reflection effects on Reynolds stresses (*Including the Wall Reflection Term* (p. 676)) for the linear pressure-strain model

Other options that are available based on your case setup include:

- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k- ε Models in the Theory Guide)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the Reynolds stress model, the following near-wall treatments are available (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

- standard wall functions
- non-equilibrium wall functions
- enhanced wall treatment

If wall boundary conditions for the Reynolds stresses from the k equation and/or wall reflection effects on Reynolds stresses are/is selected, then all the above near-wall treatments are available for selection.

If you choose the enhanced wall treatment, the following options are available:

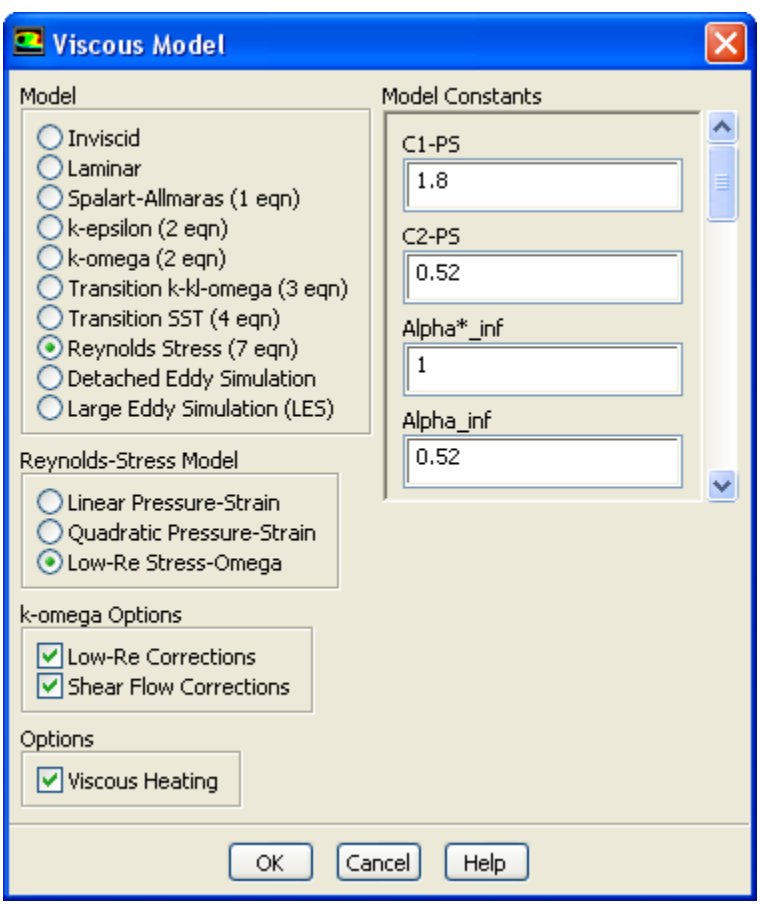
- pressure gradient effects (*Including Pressure Gradient Effects* (p. 675))
- thermal effects (*Including Thermal Effects* (p. 676))

If the quadratic pressure-strain model is selected, then you can set either the standard wall functions or the non-equilibrium wall functions.

If **Low-Re Stress-Omega** is selected, you cannot select any near-wall treatments and should use an LRN mesh. You do have the option of selecting any or all of the following $k-\omega$ options:

- low-Re corrections (*Low-Re Corrections* (p. 675))
- shear flow corrections (*Shear Flow Corrections* (p. 675))

Figure 13.10 The Viscous Model Dialog Box Displaying the Low-Re Stress-Omega Model Options



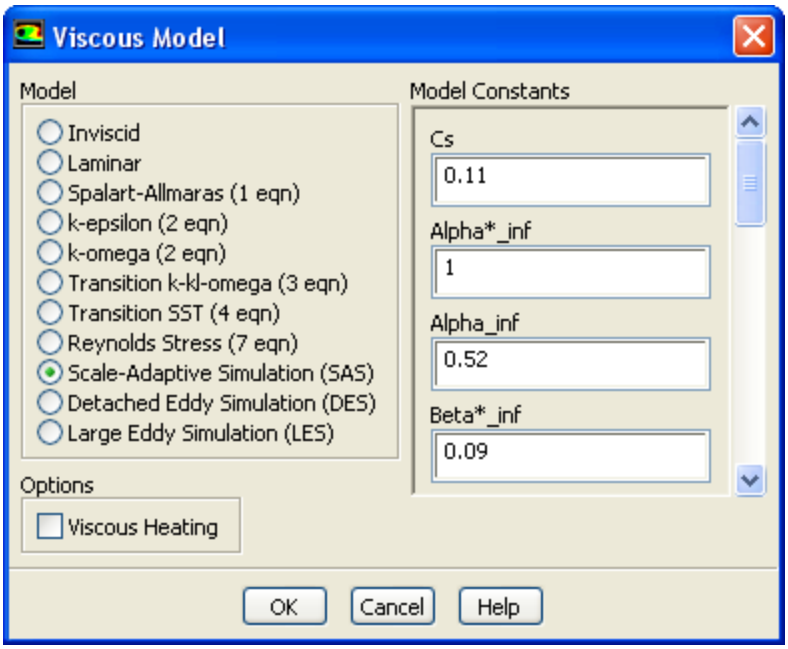
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.10. Setting Up the Scale-Adaptive Simulation (SAS) Model

The SAS turbulence model is an approach for the simulation of unsteady turbulent flows. You can enable the SAS turbulence model by selecting **Scale-Adaptive Simulation (SAS)** from the **Model** list in the **Viscous Model** dialog box.

Models → **Viscous** → **Edit...**

Figure 13.11 The Viscous Model Dialog Box Displaying the Scale-Adaptive Simulation (SAS) Model



For SAS and DES simulation setups, the **Bounded Central Differencing** scheme (available in the **Solution Methods** task page) is recommended for momentum discretization, and is the default setting.

13.11. Setting Up the Detached Eddy Simulation Model

The following submodels are available when selecting the DES model:

- Spalart-Allmaras
- Realizable k- ε
- SST k- ω

For additional information, please see the following sections:

13.11.1. Setting Up the Spalart-Allmaras DES Model
13.11.2. Setting Up the Realizable k- ε DES Model
13.11.3. Setting Up the SST k- ω DES Model

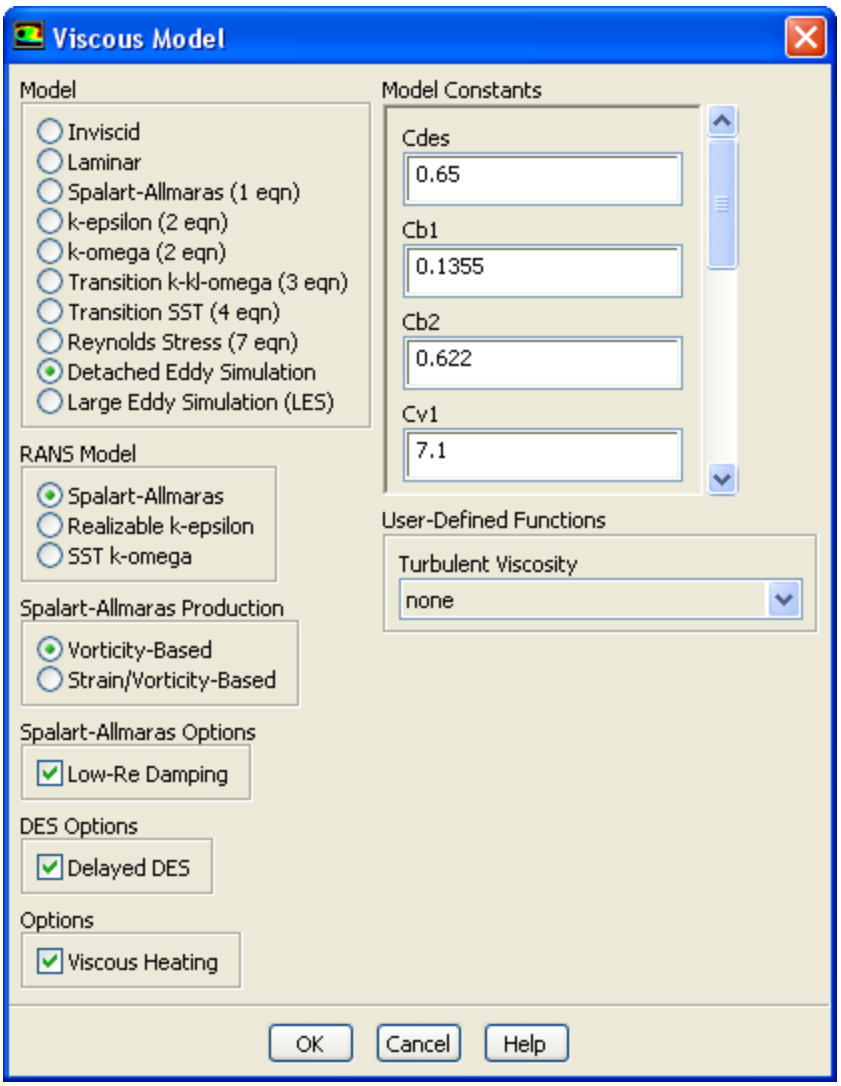
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.11.1. Setting Up the Spalart-Allmaras DES Model

ANSYS FLUENT uses Equation 4–229 (in the Theory Guide) to compute the value of the length scale $\tilde{d}$ for the Spalart-Allmaras model. By default, the empirical constant C_{des} is set to 0.65. You can change its value in the **Cdes** field under **Model Constants**. The following options are available for this model:

- vorticity-based production (*Vorticity- and Strain/Vorticity-Based Production* (p. 674))
- strain/vorticity-based production (*Vorticity- and Strain/Vorticity-Based Production* (p. 674))
- low-Re damping (*Low-Re Damping* (p. 675))
- delayed DES (*Delayed Detached Eddy Simulation (DDES)* (p. 674))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))

Figure 13.12 The Viscous Model Dialog Box Displaying the Spalart-Allmaras Detached Eddy Simulation Model Options



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

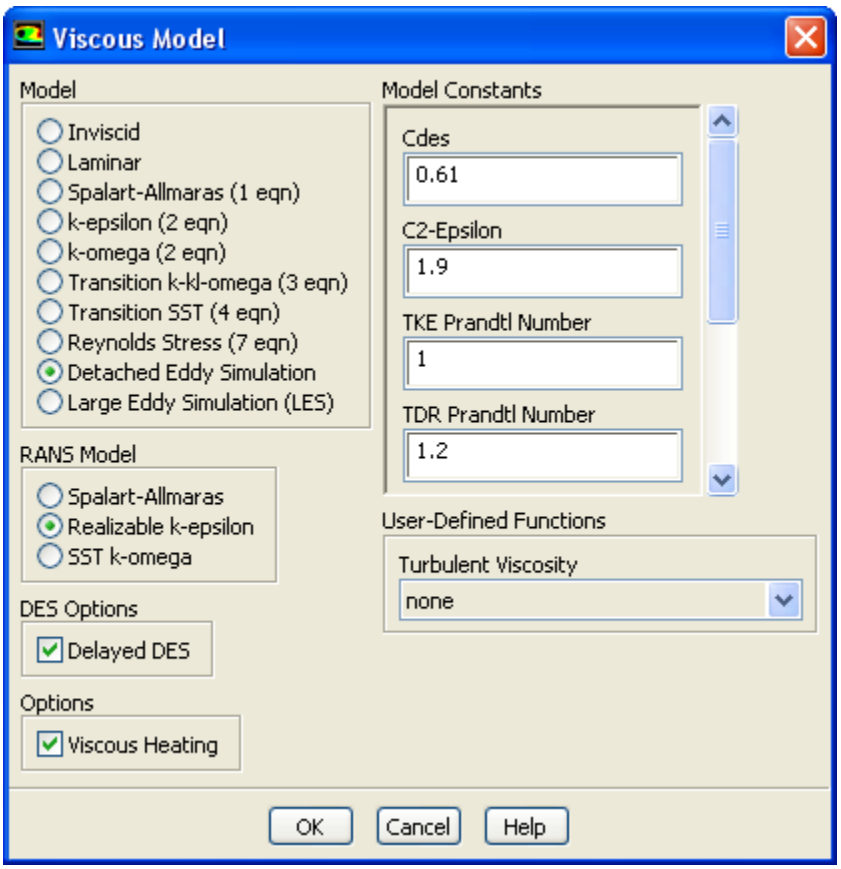
13.11.2. Setting Up the Realizable k- ε DES Model

For the Realizable k- ε submodel, the model-specific options are

- delayed DES (*Delayed Detached Eddy Simulation (DDES)* (p. 674))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))

The model constant C_{des} is set to 0.61 for the Realizable k- ε model (see Realizable k- ε Based DES Model in the Theory Guide).

Figure 13.13 The Viscous Model Dialog Box Displaying the Realizable k- ε Detached Eddy Simulation Model Options



13.11.3. Setting Up the SST k- ω DES Model

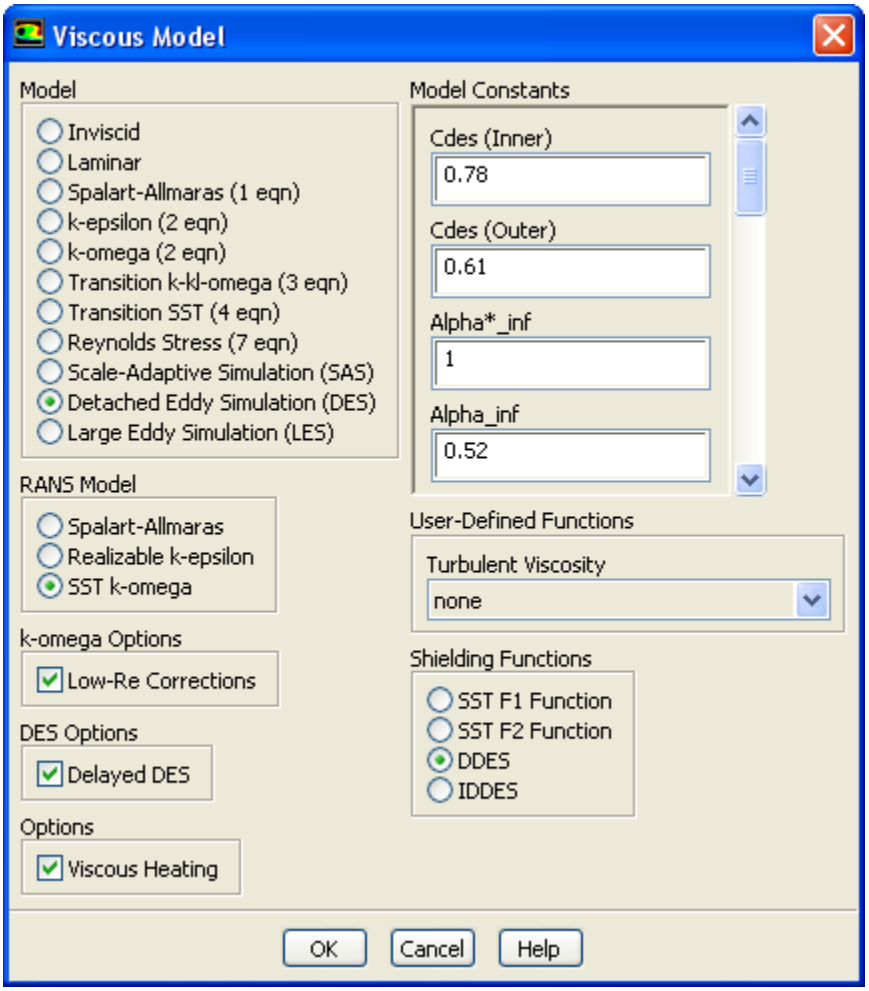
For the SST k- ω sub-model, the model-specific options that you can select are

- delayed DES (*Delayed Detached Eddy Simulation (DDES)* (p. 674))
- viscous heating (always activated for the density-based solvers) (*Including the Viscous Heating Effects* (p. 674))
- low-Re corrections k- ω option (*Low-Re Corrections* (p. 675))
- Shielding functions, F1, F2, DDES (Delayed DES) and IDDES (Improved Delayed DES) (*Shielding Functions for the SST Detached Eddy Simulation Model* (p. 678))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The model constant C_{des} is set to 0.61 for the SST k- ω RANS model (see SST k- ω Based DES Model in the Theory Guide).

Figure 13.14 The Viscous Model Dialog Box Displaying the SST k- ω Detached Eddy Simulation Model Options



Additionally, you can perform the following DES-specific functions by using the `/define/models/viscous/detached-eddy-simulation?` text command:

- Use cell volume-based LES length scale (default is to use maximum cell edge)
- Modify only the length scales that appear in the destruction term in ν_t equation (the default is to modify all length scales within the ν_t equation)

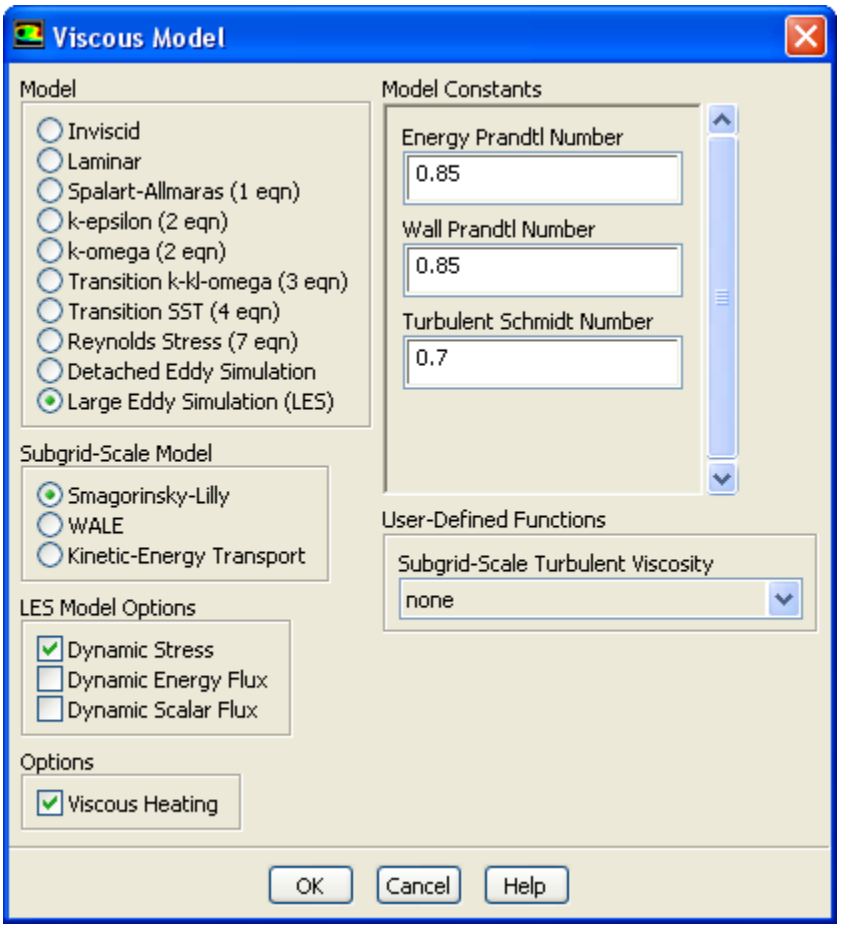
13.12. Setting Up the Large Eddy Simulation Model

If you choose the LES model, the following subgrid-scale submodels are available (*Subgrid-Scale Model* (p. 677)):

- Smagorinsky-Lilly
- WALE
- Kinetic-Energy Transport

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.15 The Viscous Model Dialog Box Displaying the Large Eddy Simulation Model Options



The LES options that are available for the Smagorinsky-Lilly are

- Dynamic Stress
- Dynamic Energy Flux (available only when the Dynamic Stress Model is enabled)
- Dynamic Scalar Flux

The LES options that are available when the Kinetic-Energy Transport submodel is selected are **Dynamic Energy Flux** and **Dynamic Scalar Flux**.

It is also possible to modify the **Model Constants**, but this is not necessary for most applications. For more information about the constants, see Spalart-Allmaras Model through Large Eddy Simulation (LES) Model (in the Theory Guide). Note that **C1-PS** and **C2-PS** are the constants C_1 and C_2 in the linear pressure-strain approximation of Equation 4–189 and Equation 4–190 (in the Theory Guide), and **C1'-PS** and **C2'-PS** are the constants C_1' and C_2' in Equation 4–191 (in the Theory Guide). **C1-SSG-PS**, **C1'-SSG-PS**, **C2-SSG-PS**, **C3-SSG-PS**, **C3'-SSG-PS**, **C4-SSG-PS**, and **C5-SSG-PS** are the constants C_1, C_1^*, C_2, C_3, C_3^*, C_4, and C_5 in the quadratic pressure-strain approximation of Equation 4–200 (in the Theory Guide).

13.13. Setting Up the Embedded Large Eddy Simulation (E-LES) Model

As described in Embedded Large Eddy Simulation (E-LES) in the Theory Guide, the Embedded Large Eddy Simulation model is used when modeling a smaller embedded LES zone within a larger RANS computational

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

domain. Recall that the interface between the upstream RANS zone and the LES zone needs to be defined (by assigning the interface to a velocity fluctuation algorithm). In addition, the interface between the LES zone and the downstream RANS zone needs to be considered.

This section describes how to set up the Embedded Large Eddy Simulation model.

1. In the **Viscous Model** dialog box, enable any RANS model (k- ε, k- ω, etc.), or you can select DES or SAS. The only RANS model not compatible with E-LES is the Spalart-Allmaras model, as a one-equation model cannot provide the required turbulent lengths scale to the interface method.

   If a RANS model is selected, the model is applied globally to the computational domain, however, values are frozen within the E-LES region. The frozen state of the E-LES zone is used to determine the flow conditions (for k- ε, k- ω, etc.) at the downstream LES-RANS zone interface. Note that this approach requires a fairy well converged global RANS solution to start with.

   If DES or SAS is used, these models are not frozen, but run in the background in the E-LES region, obtaining proper flow conditions (for k- ε, k- ω, etc.) at the downstream LES-RANS zone interface.

2. For the specified fluid cell zone, enable **LES Zone** in the **Fluid** dialog box (*Figure 13.16* (p. 672)). This enables the **Embedded LES** tab in the **Fluid** dialog box.

   Note

   When DES or SAS is used for the global model, this and the next step can, but need not, be skipped. (Keep in mind the general limitations of DES in free, wall-independent flows.) If you choose to skip these steps, then proceed with step 4.

3. In the **Embedded LES** tab, you can then specify the **Embedded Subgrid Scale Model**, and the **Momentum Spatial Discretization**.

   For the **Embedded Subgrid Scale Model**, the following subgrid-scale submodels are available (*Subgrid-Scale Model* (p. 677)):

   - Smagorinsky-Lilly
   - WALE
   - Dynamic Smagorinsky-Lilly

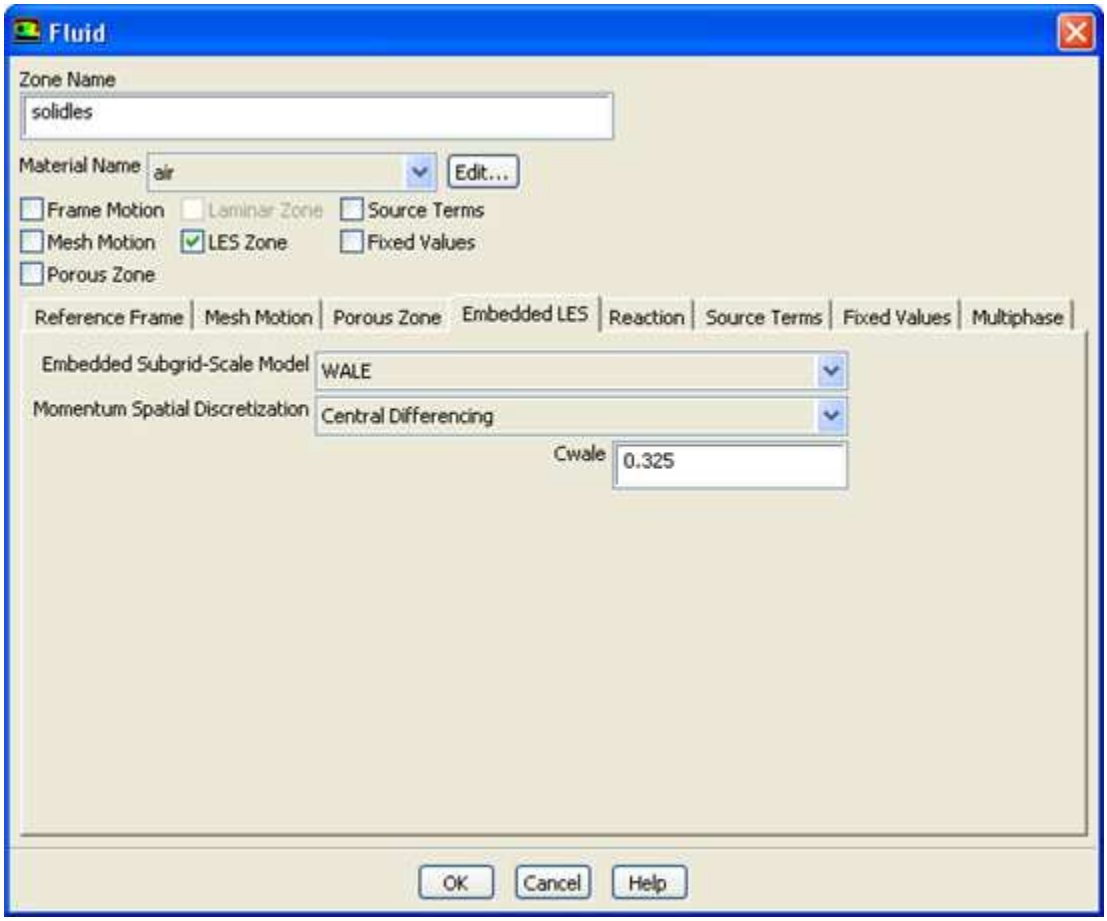
   When the Smagorinsky-Lilly model is selected, you can specify a value for C_s (see Dynamic Smagorinsky-Lilly Model in the Theory Guide for details). Likewise, when the WALE model is selected, you can specify a value for C_{wale} (or C_w, see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide for details).

   For the **Momentum Spatial Discretization**, the following options are available:

   - Bounded Central Differencing
   - Central Differencing

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.16 Specifying an E-LES Zone in the Fluid Dialog Box



4. Select an appropriate interior interface and designate it as the RANS-LES interface, by selecting the boundary zone under **Zones** in the **Boundary Conditions** task page, and assign to **rans-les-interface** for the boundary **Type**. Click **Edit...** to display the **RANS/LES Interface** dialog box (*Figure 13.17* (p. 673)), where you can assign a **Zone Name**, as well as the **Fluctuating Velocity Algorithm**, and the **Number of Vortices**. The options for the **Fluctuating Velocity Algorithm** are:

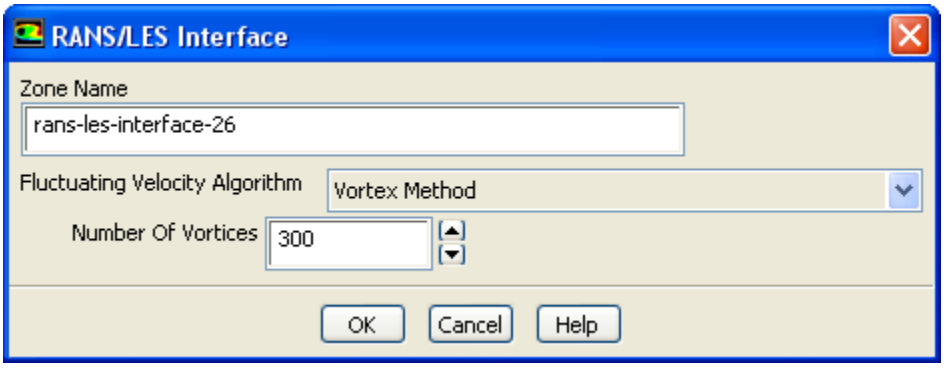
   - No Perturbations
   - Spectral Synthesizer
   - Vortex Method

   For more information about these options, please refer to Inlet Boundary Conditions for the LES Model in the Theory Guide.

   Note that the **Number of Vortices** is the amount of vortices that the selected method distributes randomly over the face zone and uses to generate turbulent fluctuations. The value should be large enough to make sure there are no spots on the face zone that are unaffected by any vortex. Large numbers may slightly increase the CPU effort, but will not impair the results.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.17 Specifying a RANS/LES Interface



If the RANS/LES interface is a non-conformal mesh interface, you can find the name of the interior zone that you want to change into a **rans-les-interface** zone in the *Mesh Interfaces Task Page* (p. 1875).

Important

It is recommended that the RANS-LES interface be situated in a region where there is no backflow.

13.14. Setup Options for all Turbulence Modeling

For more information about the various options available for the turbulence models, see Spalart-Allmaras Model through Large Eddy Simulation (LES) Model (in the Theory Guide). Instructions for activating these options are provided here.

For additional information, please see the following sections:

13.14.1. Including the Viscous Heating Effects
13.14.2. Including Turbulence Generation Due to Buoyancy
13.14.3. Vorticity- and Strain/Vorticity-Based Production
13.14.4. Delayed Detached Eddy Simulation (DDES)
13.14.5. Differential Viscosity Modification
13.14.6. Swirl Modification
13.14.7. Low-Re Corrections
13.14.8. Low-Re Damping
13.14.9. Shear Flow Corrections
13.14.10. Turbulence Damping
13.14.11. Including Pressure Gradient Effects
13.14.12. Including Thermal Effects
13.14.13. Including the Wall Reflection Term
13.14.14. Solving the k Equation to Obtain Wall Boundary Conditions
13.14.15. Quadratic Pressure-Strain Model
13.14.16. Low-Re Stress-Omega Pressure-Strain
13.14.17. Subgrid-Scale Model
13.14.18. Customizing the Turbulent Viscosity
13.14.19. Customizing the Turbulent Prandtl and Schmidt Numbers
13.14.20. Modeling Turbulence with Non-Newtonian Fluids
13.14.21. Shielding Functions for the SST Detached Eddy Simulation Model

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.14.1. Including the Viscous Heating Effects

For information about including viscous heating effects in your model, see Inclusion of the Viscous Dissipation Terms in the Theory Guide and *Solving Heat Transfer Problems* (p. 691).

13.14.2. Including Turbulence Generation Due to Buoyancy

If you specify a non-zero gravity force (in the *Operating Conditions Dialog Box* (p. 1806)), and you are modeling a non-isothermal flow, the generation of turbulent kinetic energy due to buoyancy (G_b in Equation 4–33 is, by default, always included in the k equation. However, ANSYS FLUENT does not, by default, include the buoyancy effects on ε.

To include the buoyancy effects on ε, you must turn on the **Full Buoyancy Effects** option under **Options** in the *Viscous Model Dialog Box* (p. 1645).

This option is available for all three k- ε models and for the RSM.

13.14.3. Vorticity- and Strain/Vorticity-Based Production

For the Spalart-Allmaras model, you can choose either **Vorticity-Based Production** or **Strain/Vorticity-Based Production** under **Spalart-Allmaras Production** in the *Viscous Model Dialog Box* (p. 1645). If you choose **Vorticity-Based Production**, ANSYS FLUENT will compute the value of the deformation tensor S using Equation 4–22 (in the Theory Guide); if you choose **Strain/Vorticity-Based Production**, it uses Equation 4–24. (in the Theory Guide)

(These options will not appear unless you have activated the Spalart-Allmaras model.)

13.14.4. Delayed Detached Eddy Simulation (DDES)

The **Delayed DES** option is recommended when using the DES model. This option preserves the RANS model throughout the boundary layer. For more information, see Detached Eddy Simulation (DES) in the Theory Guide.

13.14.5. Differential Viscosity Modification

The RNG turbulence model in ANSYS FLUENT has an option of using a differential formula for the effective viscosity μ_{eff} (Equation 4–38 in the Theory Guide) to account for the low-Reynolds-number effects. To enable this option, the **Differential Viscosity Model** option under **RNG Options** in the *Viscous Model Dialog Box* (p. 1645) needs to be enabled.

Important

This option appears when you have activated the RNG k- ε model.

13.14.6. Swirl Modification

After you have chosen the RNG model, the swirl modification takes effect, by default, for all three-dimensional flows and axisymmetric flows with swirl. The default swirl constant (α_s in Equation 4–40 in the Theory Guide) is set to 0.07, which works well for weakly to moderately swirling flows. However, for strongly swirling flows, you may need to use a larger swirl constant.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change the value of the swirl constant, you must first enable the **Swirl Dominated Flow** option under **RNG Options** in the *Viscous Model Dialog Box* (p. 1645).

Important

This option will not appear unless you have activated the RNG k- ε model.

13.14.7. Low-Re Corrections

If either of the k- ω models are used, you may enable a low-Reynolds-number correction to the turbulent viscosity by enabling the **Low-Re Corrections** option under **k-omega Options** in the **Viscous Model** dialog box. By default, this option is not enabled, and the damping coefficient (α^* in Equation 4–68 in the Theory Guide) is equal to 1.

13.14.8. Low-Re Damping

If either of the Spalart-Allmaras or the Spalart-Allmaras Detached Eddy Simulation models are used, you may enable a low-Reynolds damping to the turbulent viscosity by enabling the **Low-Re Damping** option under **Spalart-Allmaras Options** in the **Viscous Model** dialog box. This is required if you run low viscous sublayer meshes with $y^+ < 3$.

13.14.9. Shear Flow Corrections

In the standard k- ω model, you also have the option of including corrections to improve the accuracy in predicting free shear flows. The **Shear Flow Corrections** option under the **k-omega Options** is enabled by default in the **Viscous Model** dialog box, as these corrections are included in the standard k- ω model [*102*] (p. 2214). When this option is enabled, ANSYS FLUENT will calculate f_β^* using Equation 4–78 (in the Theory Guide) and f_β using Equation 4–85 (in the Theory Guide). If this option is disabled, f_β^* and f_β will be set equal to 1.

13.14.10. Turbulence Damping

In the standard or SST k- ω model, you have the option of including turbulence damping, which is required for the accurate modeling of the interfacial area. The **Turbulence Damping** option under **k-omega Options** is available for the VOF and Mixture models and also available with the Eulerian multiphase model when using the immiscible fluid model. When this option is enabled, you can set the **Damping Factor**, which by default is set to 10. For a theoretical discussion about turbulence damping, please refer to Turbulence Damping.

13.14.11. Including Pressure Gradient Effects

If the enhanced wall treatment is used, you may include the effects of pressure gradients by enabling the **Pressure Gradient Effects** option under the **Enhanced Wall Treatment Options**. When this option is enabled, ANSYS FLUENT will include the coefficient α in Equation 4–305 (in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.14.12. Including Thermal Effects

If the enhanced wall treatment is used, you may include thermal effects by enabling the **Thermal Effects** option under **Enhanced Wall Treatment Options**. When this option is enabled, ANSYS FLUENT will include the coefficient β in Equation 4–305 (in the Theory Guide). γ will also be included in Equation 4–305 when the **Thermal Effects** option is enabled if the ideal gas law is selected for the fluid density in the **Create/Edit Materials** dialog box.

13.14.13. Including the Wall Reflection Term

If the RSM is used with the default model for pressure strain, ANSYS FLUENT will, by default, include the wall-reflection effects in the pressure-strain term. That is, ANSYS FLUENT will calculate $\phi_{ij,w}$ using Equation 4–191 (in the Theory Guide) and include it in Equation 4–188 (in the Theory Guide). Note that wall-reflection effects are not included if you have selected the quadratic pressure-strain model.

Important

The empirical constants and the function f used in the calculation of $\phi_{ij,w}$ are calibrated for simple canonical flows such as channel flows and flat-plate boundary layers involving a single wall. If the flow involves multiple walls and the wall has significant curvature (e.g., an axisymmetric pipe or curvilinear duct), the inclusion of the wall-reflection term in Equation 4–191 (in the Theory Guide) may not improve the accuracy of the RSM predictions. In such cases, you can disable the wall-reflection effects by turning off the **Wall Reflection Effects** under **Reynolds-Stress Options** in the *Viscous Model Dialog Box* (p. 1645).

13.14.14. Solving the k Equation to Obtain Wall Boundary Conditions

In the RSM, ANSYS FLUENT, by default, uses the explicit setting of boundary conditions for the Reynolds stresses near the walls, with the values computed with Equation 4–218 (in the Theory Guide). The turbulent kinetic energy, k, is calculated by solving the k equation obtained by summing Equation 4–185 (in the Theory Guide) for normal stresses. To disable this option and use the wall boundary conditions given in Equation 4–219 (in the Theory Guide), turn off the **Wall BC from k Equation** under the **Reynolds-Stress Options** in the **Viscous Model** dialog box.

Important

This option will not appear unless you have activated the **Reynolds Stress** model.

13.14.15. Quadratic Pressure-Strain Model

To use the quadratic pressure-strain model described in Quadratic Pressure-Strain Model (in the Theory Guide), enable the **Quadratic Pressure-Strain Model** option under **Reynolds-Stress Options** in the *Viscous Model Dialog Box* (p. 1645). (This option will not appear unless you have activated the RSM.) The following options are not available when the **Quadratic Pressure-Strain Model** is enabled:

- **Wall Reflection Effects** under **Reynolds-Stress Options**
- **Enhanced Wall Treatment** under **Near-Wall Treatment**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.14.16. Low-Re Stress-Omega Pressure-Strain

To use the Low-Reynold-Number Stress-Omega option described in Low-Re Stress-Omega Model, turn on the **Low-Re Stress-Omega** option under **Reynolds-Stress Model** in the *Viscous Model Dialog Box* (p. 1645). (This option will not appear unless you have activated the RSM.) The following options are not available when the **Low-Re Stress-Omega** option is enabled:

- **Wall BC from k Equation** under **Reynolds-Stress Options**
- **Wall Reflection Effects** under **Reynolds-Stress Options**
- **Standard Wall Functions** under **Near-Wall Treatment**
- **Non-Equilibrium Wall Functions** under **Near-Wall Treatment**
- **Enhanced Wall Treatment** under **Near-Wall Treatment**

Instead, the following options have to be set:

- **Low-Re Corrections** under **k-omega Options**
- **Shear Flow Corrections** under **k-omega Options**

13.14.17. Subgrid-Scale Model

If you have selected the **Large Eddy Simulation** model, you will be able to choose one of the subgrid-scale models described in Subgrid-Scale Models (in the Theory Guide). You can choose from the **Smagorinsky-Lilly**, **WALE**, or **Kinetic-Energy Transport** subgrid-scale models. Note that **Dynamic Stress** is an option available with the **Smagorinsky-Lilly** model, while the **Kinetic-Energy Transport** model is always run as a dynamic model.

Important

These options will not appear unless you have activated the LES model.

13.14.18. Customizing the Turbulent Viscosity

If you are using the Spalart-Allmaras, k- ε, k- ω, DES, or LES models, a UDF can be used to customize the turbulent viscosity. This option will enable you to modify μ_t in the case of the Spalart-Allmaras, k- ε, and k- ω models, and incorporate completely new subgrid models in the case of the LES model. More information about UDFs can be found in the UDF Manual.

In the *Viscous Model Dialog Box* (p. 1645), under **User-Defined Functions**, select the appropriate user-defined function in the **Turbulent Viscosity** drop-down list. For the LES model, select the appropriate UDF in the **Subgrid-Scale Turbulent Viscosity** drop-down list.

13.14.19. Customizing the Turbulent Prandtl and Schmidt Numbers

If you are using the standard or realizable k- ε model or the standard k- ω model, a UDF can be used to customize the turbulent Prandtl and Schmidt numbers. This option will allow you to calculate σ_k and either σ_ε or σ_ω (depending on the choice of either k- ε or k- ω model) by using a UDF. You will also be able to calculate the value of the energy turbulent Prandtl number (Pr_t in Equation 4–57 in the Theory Guide) and the turbulent Prandtl number at the wall (Pr_t in Equation 4–278 in the Theory Guide) in this way. More information about UDFs can be found in the UDF Manual.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the *Viscous Model Dialog Box* (p. 1645), under **User-Defined Functions**, select the appropriate UDF from the drop-down lists under **Prandtl and Schmidt Numbers**. Options include: **TKE Prandtl Number**, **TDR Prandtl Number** (k- ε models only), **SDR Prandtl Number** (k- ω model only), **Energy Prandtl Number**, **Wall Prandtl Number**, and **Turbulent Schmidt Number**.

13.14.20. Modeling Turbulence with Non-Newtonian Fluids

If the turbulent flow involves non-Newtonian fluids, you can use the `define/models/ viscous/turbulence-expert/turb-non-newtonian?` text command to enable the selection of non-Newtonian options for the material viscosity. See *Viscosity for Non-Newtonian Fluids* (p. 413) for details about these options.

13.14.21. Shielding Functions for the SST Detached Eddy Simulation Model

The SST shielding (or blending) functions, F1 and F2, are defined in Equation 4–102 and Equation 4–104 in the Theory Guide, respectively. The default is F2, which is more conservative than F1. In addition, the SST shielding functions, DDES (Delayed DES) and IDDES (Improved Delayed DES), are described in SST k- ω Based DES Model and Improved Delayed Detached Eddy Simulation (IDDES) in the Theory Guide, respectively, and can also be used with the E-LES model (*Setting Up the Embedded Large Eddy Simulation (E-LES) Model* (p. 670)).

13.15. Defining Turbulence Boundary Conditions

For additional information, please see the following sections:

13.15.1. The Spalart-Allmaras Model
13.15.2. k- ε Models and k- ω Models
13.15.3. Reynolds Stress Model
13.15.4. Large Eddy Simulation Model

13.15.1. The Spalart-Allmaras Model

When you are modeling turbulent flows in ANSYS FLUENT using the Spalart-Allmaras model, you must provide the boundary conditions for $\tilde{\nu}$ in addition to other mean solution variables. The boundary conditions for $\tilde{\nu}$ at the walls are internally taken care of by ANSYS FLUENT, which obviates the need for your inputs. The boundary condition input for $\tilde{\nu}$, which you must enter in ANSYS FLUENT, is the one at inlet boundaries (velocity inlet, pressure inlet, etc.). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

You may want to include the effects of the wall roughness on selected wall boundaries. In such cases, you can specify the roughness parameters (roughness height and roughness constant) in the dialog boxes of the corresponding wall boundaries (see *Setting the Roughness Parameters* (p. 308)).

13.15.2. k- ε Models and k- ω Models

When you are modeling turbulent flows in ANSYS FLUENT using one of the k- ε models or one of the k- ω models, you must provide the boundary conditions for k and ε (or k and ω) in addition to other mean solution variables. The boundary conditions for k and ε (or k and ω) at the walls are internally taken care of by ANSYS FLUENT, which obviates the need for your inputs. The boundary condition inputs for k and ε (or k and ω), which you must enter in ANSYS FLUENT, are the ones at inlet boundaries (velocity inlet, pressure inlet, etc.). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See *Determining Turbulence Parameters* (p. 250) for details about specifying the boundary conditions for k and ε (or k and ω) at the inlets.

You may want to include the effects of the wall roughness on selected wall boundaries. In such cases, you can specify the roughness parameters (roughness height and roughness constant) in the dialog boxes for the corresponding wall boundaries (see *Setting the Roughness Parameters* (p. 308)).

Additionally, you can control whether or not to set the turbulent viscosity to zero within a laminar zone. If the fluid zone in question is laminar, the text command `define/boundary-conditions/fluid` will contain an option called `Set Turbulent Viscosity to zero within laminar zone?`. By setting this option to `yes`, ANSYS FLUENT will set both the production term in the turbulence transport equation and μ_t to zero. In contrast, when the **Laminar Zone** option is enabled in a **Fluid** cell zone condition dialog box, only the production term is set to zero. See *Specifying a Laminar Zone* (p. 216) for details about laminar zones.

Important

Note that the laminar zone feature is also available for the Spalart-Allmaras and RSM models.

13.15.3. Reynolds Stress Model

The specification of turbulent boundary conditions for the RSM is the same as for the other turbulence models for all boundaries except at boundaries where flow enters the domain. Additional input methods are available for these boundaries and are described here.

When you choose to use the RSM, the default inlet boundary condition inputs required are identical to those required when the k- ε model is active. You can input the turbulence quantities using any of the turbulence specification methods described in *Determining Turbulence Parameters* (p. 250). ANSYS FLUENT then uses the specified turbulence quantities to derive the Reynolds stresses at the inlet from the assumption of isotropy of turbulence:

$$\overline{u_i'^2} = \frac{2}{3}k \quad (i=1,2,3) \tag{13–6}$$

$$\overline{u_i' u_j'} = 0.0 \tag{13–7}$$

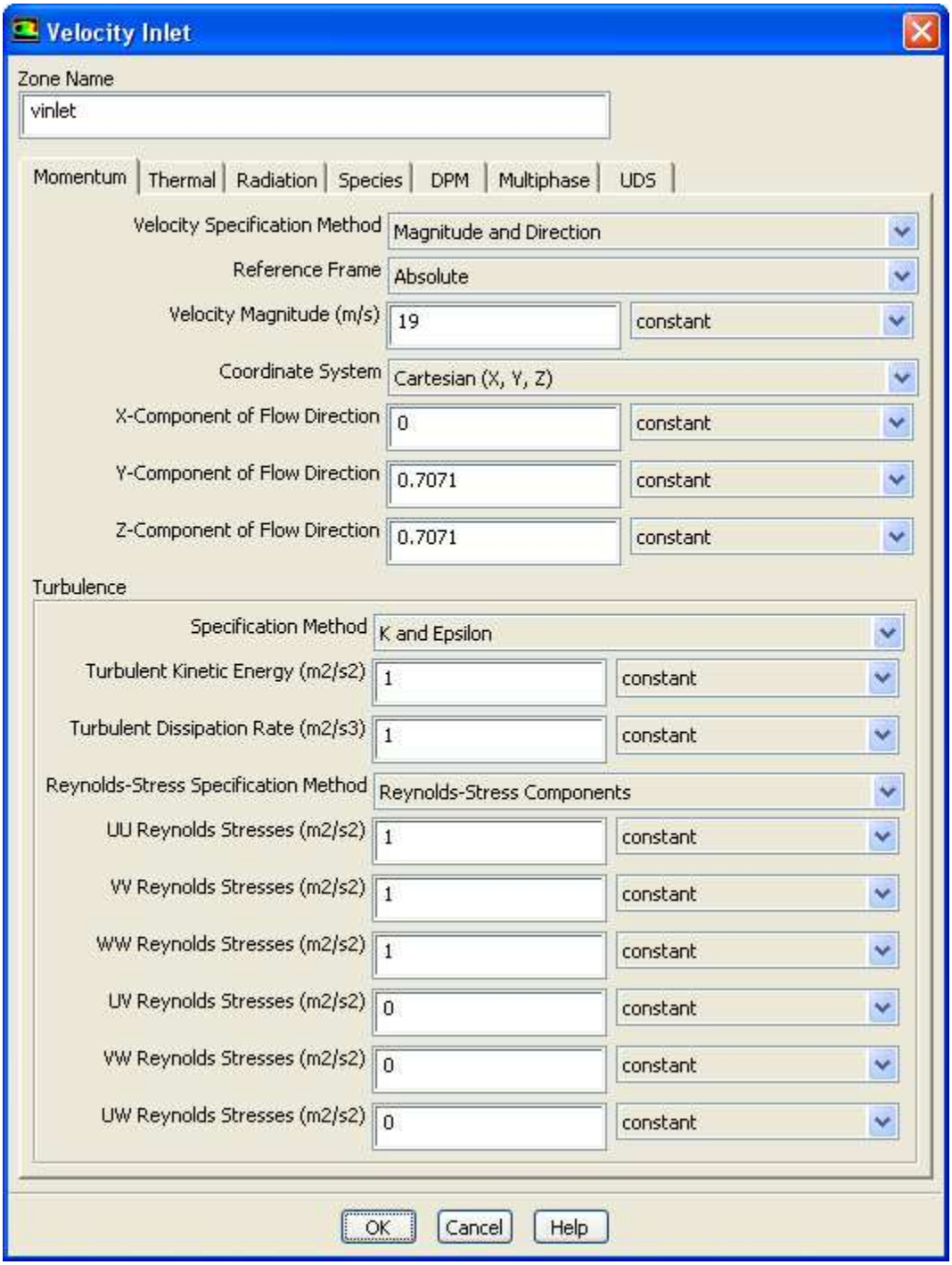
where $\overline{u_i'^2}$ is the normal Reynolds stress component in each direction. The boundary condition for ε is determined in the same manner as for the k- ε turbulence models (see *Determining Turbulence Parameters* (p. 250)). To use this method, you will select **K or Turbulence Intensity** as the **Reynolds-Stress Specification Method** in the appropriate boundary condition dialog box.

Alternately, you can directly specify the Reynolds stresses by selecting the **Reynolds-Stress Components** as the **Reynolds-Stress Specification Method** in the boundary condition dialog box. When this option is selected, you should input the Reynolds stresses directly.

You can set the Reynolds stresses by using constant values, profile functions of coordinates (see *Profiles* (p. 366)), or user-defined functions (in the UDF Manual).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.18 Specifying Inlet Boundary Conditions for the Reynolds Stresses



13.15.4. Large Eddy Simulation Model

It is possible to specify the magnitude of random fluctuations of the velocity components at an inlet only if the velocity inlet boundary condition is selected. In this case, you must specify a **Turbulence Intensity** that determines the magnitude of the random perturbations on individual mean velocity components as described in Inlet Boundary Conditions for the LES Model (in the Theory Guide). For all boundary types other than velocity inlets, the boundary conditions for LES remain the same as for laminar flows.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.16. Providing an Initial Guess for k and ε (or k and ω)

For flows using one of the k- ε models, one of the k- ω models, or the RSM, the converged solutions or (for unsteady calculations) the solutions after a sufficiently long time has elapsed should be independent of the initial values for k and ε (or k and ω). For better convergence, however, it is beneficial to use a reasonable initial guess for k and ε (or k and ω).

In general, it is recommended that you start from a fully-developed state of turbulence. When you use the enhanced wall treatment for the k- ε models or the RSM, it is critically important to specify fully-developed turbulence fields. Guidelines are provided below.

- If you were able to specify reasonable boundary conditions at the inlet, it may be a good idea to compute the initial values for k and ε (or k and ω) in the whole domain from these boundary values. (See *Initializing the Solution* (p. 1239) for details.)
- For more complex flows (e.g., flows with multiple inlets with different conditions) it may be better to specify the initial values in terms of turbulence intensity. 5–10% is enough to represent fully-developed turbulence. The values of k can then be computed from the turbulence intensity and the characteristic mean velocity magnitude of your problem ($k = 1.5\left(Iu_{avg}\right)^2$).

  You should specify an initial guess for ε so that the resulting eddy viscosity ($C_\mu \frac{k^2}{\varepsilon}$) is sufficiently large in comparison to the molecular viscosity. In fully-developed turbulence, the turbulent viscosity is roughly two orders of magnitude larger than the molecular viscosity. From this, you can compute ε.

$$\varepsilon = C_\mu \frac{k^2}{\mu_T} = C_\mu \frac{k^2}{\frac{\mu_T}{\mu}\mu} \tag{13–8}$$

where $\frac{\mu_T}{\mu}$ is the turbulent viscosity ratio which can be prescribed at the inlet and then used for the domain initialization.

Note that, for the RSM, Reynolds stresses are initialized automatically using *Equation 13–6* (p. 679) and *Equation 13–7* (p. 679).

13.17. Solution Strategies for Turbulent Flow Simulations

Compared to laminar flows, simulations of turbulent flows are more challenging in many ways. For the Reynolds-averaged approach, additional equations are solved for the turbulence quantities. Since the equations for mean quantities and the turbulent quantities (μ_t, k, ε, ω, or the Reynolds stresses) are strongly coupled in a highly non-linear fashion, it takes more computational effort to obtain a converged turbulent solution than to obtain a converged laminar solution. The LES model, while embodying a simpler, algebraic model for the subgrid-scale viscosity, requires a transient solution on a very fine mesh.

The fidelity of the results for turbulent flows is largely determined by the turbulence model being used. Here are some guidelines that can enhance the quality of your turbulent flow simulations.

For additional information, please see the following sections:

13.17.1. Mesh Generation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.17.2. Accuracy
13.17.3. Convergence
13.17.4. RSM-Specific Solution Strategies
13.17.5. LES-Specific Solution Strategies

13.17.1. Mesh Generation

The following are suggestions to follow when generating the mesh for use in your turbulent flow simulation:

- First imagine the flow under consideration, then identify the main flow features expected in the flow using your physical intuition or any data for a similar flow situation. Generate a mesh that can resolve the major features that you expect.
- If the flow is wall-bounded, and the wall is expected to affect the flow significantly, then take additional care when generating the mesh. You should avoid using a mesh that is too fine (for the wall-function approach) or too coarse (for the enhanced wall treatment approach). See *Grid Resolution RANS Models* (p. 647) and *Wall Boundary Layers* (p. 651) for details.

13.17.2. Accuracy

The suggestions below are provided to help you obtain better accuracy in your results:

- Use the turbulence model that is better suited for the salient features you expect to see in the flow (see *Choosing a Turbulence Model* (p. 644)).
- Because the mean quantities have larger gradients in turbulent flows than in laminar flows, it is recommended that you use high-order schemes for the convection terms. This is especially true if you employ a triangular or tetrahedral mesh. Note that excessive numerical diffusion adversely affects the solution accuracy, even with the most elaborate turbulence model.
- In some flow situations involving inlet boundaries, the flow downstream of the inlet is dictated by the boundary conditions at the inlet. In such cases, you should exercise care to make sure that reasonably realistic boundary values are specified.

13.17.3. Convergence

The suggestions below are provided to help you enhance convergence for turbulent flow calculations:

- Starting with excessively crude initial guesses for mean and turbulence quantities may cause the solution to diverge. A safe approach is to start your calculation using conservative (small) under-relaxation parameters and (for the density-based solvers) a conservative Courant number, and increase them gradually as the iterations proceed and the solution begins to settle down.
- It is also helpful for faster convergence to start with reasonable initial guesses for the k and ε (or k and ω) fields. Particularly when the enhanced wall treatment is used, it is important to start with a sufficiently developed turbulence field, as recommended in *Providing an Initial Guess for k and ε (or k and ω)* (p. 681), to avoid the need for an excessive number of iterations to develop the turbulence field.
- When you are using the RNG k- ε model, an approach that might help you achieve better convergence is to obtain a solution with the standard k- ε model before switching to the RNG model. Due to the additional non-linearities in the RNG model, lower under-relaxation factors and (for the density-based solvers) a lower Courant number might also be necessary.

Note that when you use the enhanced wall treatment, you may sometimes find during the calculation that the residual for ε is reported to be zero. This happens when your flow is such that Re_y is less than 200 in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the entire flow domain, and ε is obtained from the algebraic formula (Equation 4–299 in the Theory Guide) instead of from its transport equation.

13.17.4. RSM-Specific Solution Strategies

Using the RSM creates a high degree of coupling between the momentum equations and the turbulent stresses in the flow, and thus the calculation can be more prone to stability and convergence difficulties than with the k- ε models. When you use the RSM, therefore, you may need to adopt special solution strategies in order to obtain a converged solution. The following strategies are generally recommended:

- Begin the calculations using the standard k- ε model. Turn on the RSM and use the k- ε solution data as a starting point for the RSM calculation.
- Use low under-relaxation factors (0.2 to 0.3) and (for the density-based solvers) a low Courant number for highly swirling flows or highly complex flows. In these cases, you may need to reduce the under-relaxation factors both for the velocities and for all of the stresses.

Instructions for setting these solution parameters are provided below. If you are applying the RSM to prediction of a highly swirling flow, you will want to consider the solution strategies discussed in *Swirling and Rotating Flows* (p. 497) as well.

13.17.4.1. Under-Relaxation of the Reynolds Stresses

ANSYS FLUENT applies under-relaxation to the Reynolds stresses. You can set under-relaxation factors using the *Solution Controls Task Page* (p. 1902).

Solution Controls

The default settings of 0.5 are recommended for most cases. You may be able to increase these settings and speed up the convergence when the RSM solution begins to converge.

In some situations, when poor convergence is observed one might facilitate the convergence rates by modifying some of the under-relaxation values whilst leaving the others unchanged. This might be a more successful approach than the simple scaling of all under-relaxation values.

13.17.4.2. Disabling Calculation Updates of the Reynolds Stresses

In some instances, you may wish to let the current Reynolds stress field remain fixed, skipping the solution of the Reynolds transport equations while solving the other transport equations. You can activate/deactivate all Reynolds stress equations in the **Equations** dialog box, accessed from the *Solution Controls Task Page* (p. 1902).

Solution Controls $\rightarrow$ **Equations...**

13.17.4.3. Residual Reporting for the RSM

When you use the RSM for turbulence, ANSYS FLUENT reports the equation residuals for the individual Reynolds stress transport equations. You can apply the usual convergence criteria to the Reynolds stress residuals: normalized residuals in the range of 10^{-3} usually indicate a practically-converged solution. However, you may need to apply tighter convergence criteria (below 10^{-4}) to ensure full convergence.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.17.5. LES-Specific Solution Strategies

Large eddy simulation involves running a transient solution from some initial condition, on an appropriately fine mesh, using an appropriate time step size. The solution must be run long enough to become independent of the initial condition and to enable the statistics of the flow field to be determined.

The following are suggestions to follow when running a large eddy simulation:

1. Start by running a steady state flow simulation using a Reynolds-averaged turbulence model such as standard k- ε, k- ω, or even RSM. Run until the flow field is reasonably converged and then use the `solve/initialize/init-instantaneous-vel` text command to generate the instantaneous velocity field out of the steady-state RANS results. This command must be executed before LES is enabled. This option is available for all RANS-based models and it will create a much more realistic initial field for the LES run. Additionally, it will help in reducing the time needed for the LES simulation to reach a statistically stable mode. This step is optional.
2. When you enable LES, ANSYS FLUENT will automatically turn on the unsteady solver option and choose the second-order implicit formulation. You will need to set the appropriate time step size and all the needed solution parameters. (See *User Inputs for Time-Dependent Problems* (p. 1256) for guidelines on setting solution parameters for transient calculations in general.) The bounded central differencing spatial discretization scheme will be automatically enabled for momentum equations. Both the bounded central-differencing and pure central-differencing schemes are available for all equations when running LES simulations.
3. Run LES until the flow becomes statistically steady. The best way to see if the flow is fully developed and statistically steady is to monitor forces and solution variables (e.g., velocity components or pressure) at selected locations in the flow.
4. Zero out the initial statistics using the `solve/initialize/init-flow-statistics` text command. Before you restart the solution, enable **Data Sampling for Time Statistics** in the *Run Calculation Task Page* (p. 1955), as described in *User Inputs for Time-Dependent Problems* (p. 1256). With this option enabled, ANSYS FLUENT will gather data for time statistics while performing a large eddy simulation. You can set the **Sampling Interval** such that **Data Sampling for Time Statistics** can be performed at the specified frequency. When **Data Sampling for Time Statistics** is enabled, the statistics collected at each sampling interval can be postprocessed and you can then view both the mean and the root-mean-square (RMS) values in ANSYS FLUENT.
5. Continue until you get statistically stable data. The duration of the simulation can be determined beforehand by estimating the mean flow residence time in the solution domain (L/U, where L is the characteristic length of the solution domain and U is a characteristic mean flow velocity). The simulation should be run for at least a few mean flow residence times.

Instructions for setting the solution parameters for LES are provided below.

13.17.5.1. Temporal Discretization

ANSYS FLUENT provides both first-order and second-order temporal discretizations. For LES, the second-order discretization is recommended.

Solution Methods

13.17.5.2. Spatial Discretization

Overly diffusive schemes such as the first-order upwind or power law scheme should be avoided, because they may unduly damp out the energy of the resolved eddies. The central-differencing based schemes are

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recommended for all equations when you use the LES model. ANSYS FLUENT provides two central-differencing based schemes: *pure* central-differencing and *bounded* central-differencing. The bounded scheme is the default option when you select LES or DES.

Solution Methods

13.18. Postprocessing for Turbulent Flows

ANSYS FLUENT provides postprocessing options for displaying, plotting, and reporting various turbulence quantities, which include the main solution variables and other auxiliary quantities.

Turbulence quantities that can be reported for the Spalart-Allmaras model are as follows:

- **Modified Turbulent Viscosity**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**

Turbulence quantities that can be reported for the k- ε models are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Wall Ystar**
- **Turbulent Reynolds Number (Re_y)** (only when the enhanced wall treatment is used for the near-wall treatment)

Turbulence quantities that can be reported for the k- ω models are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Specific Dissipation Rate (Omega)**
- **Production of k**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**

Turbulence quantities that can be reported for the transition k- kl- ω model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Laminar Kinetic Energy**
- **Total Fluctuation Energy**
- **Turbulent Intensity**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Production of laminar k**
- **Turbulent Viscosity**
- **Turbulent Viscosity (large-scale)**
- **Turbulent Viscosity (small-scale)**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**

Turbulence quantities that can be reported for the transition SST model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Intermittency**
- **Intermittency Effective**
- **Momentum Thickness Re**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**

Turbulence quantities that can be reported for the RSM are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **UU Reynolds Stress**
- **VV Reynolds Stress**
- **WW Reynolds Stress**
- **UV Reynolds Stress**
- **VW Reynolds Stress**
- **UW Reynolds Stress**
- **Turbulent Dissipation Rate (Epsilon)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**

Turbulence quantities that can be reported for the SAS model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Wall Ystar**
- **Normalized Q criterion**

Turbulence quantities that can be reported for the DES model are as follows:

- **Modified Turbulent Viscosity**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**
- **Relative Length Scale (DES)**
- **Normalized Q criterion**

Turbulence quantities that can be reported for the LES model are as follows:

- **Turbulence Kinetic Energy**
- **Turbulence Intensity**
- **Subgrid Kinetic Energy**
- **Production of k**
- **Subgrid Turbulent Viscosity**
- **Subgrid Effective Viscosity**
- **Subgrid Turbulent Viscosity Ratio**
- **Subgrid Filter Length**
- **Subgrid Test-Filter Length**
- **Subgrid Dissipation Rate**
- **Subgrid Dynamic Viscosity Const**
- **Subgrid Dynamic Prandtl Number**
- **Subgrid Dynamic Sc of Species**
- **Subtest Kinetic Energy**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Normalized Q criterion**

All of these variables can be found in the **Turbulence...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See *Field Function Definitions* (p. 1529) for their definitions.

For additional information, please see the following sections:
13.18.1. Custom Field Functions for Turbulence

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.18.2. Postprocessing Turbulent Flow Statistics
13.18.3. Troubleshooting

13.18.1. Custom Field Functions for Turbulence

In addition to the quantities listed in *Postprocessing for Turbulent Flows* (p. 685), above, you can define your own turbulence quantities using the *Custom Field Function Calculator Dialog Box* (p. 2106).

Define → **Custom Field Functions...**

The following functions may be useful:

- the ratio of production of k to its dissipation ($G_k/\rho\varepsilon$)
- the ratio of the mean flow to turbulent time scale, η ($\equiv Sk/\varepsilon$)
- the Reynolds stresses derived from the Boussinesq formula (e.g., $-\overline{uv}=\nu_t\frac{\partial u}{\partial y}$)

13.18.2. Postprocessing Turbulent Flow Statistics

As described in Large Eddy Simulation (LES) Model (in the Theory Guide), LES involves the solution of a transient flow field, but it is the mean flow quantities that are of interest from an engineering standpoint.

For all other turbulent flow, if **Data Sampling for Time Statistics** is enabled in the *Run Calculation Task Page* (p. 1955), ANSYS FLUENT gathers data for time statistics while performing the simulation. The statistics that ANSYS FLUENT collects at each sampling interval (which consists of the mean and the root-mean-square (RMS) values) can be postprocessed by selecting **Unsteady Statistics...** in any of the postprocessing dialog boxes. You can view several variables that include, but are not limited to, shear stresses (**Resolved UV/UW/VW Reynolds Stress**), flow heat fluxes (**Resolved UT/VT/WT Heat Flux**), and species statistics (**RMS Mass Fraction of** species and **Mean Mass Fraction** of species). If you select **Unsteady Wall Statistics...** in any of the postprocessing dialog boxes, you can view wall statistics such as **Mean Pressure Coefficient**, **Mean Wall Shear Stress**, **Mean X-Wall Shear Stress**, **Mean Y-Wall Shear Stress**, **Mean Z-Wall Shear Stress**, **Mean Skin Friction Coefficient**, **Mean Surface Heat Flux**, **Mean Surface Heat Transfer Coef.**, **Mean Surface Nusselt Number**, **Mean Surface Stanton Number**. Note that these quantities are only the statistical evaluation of sampled solution data, and do not contain any kind of modeled turbulent fluctuations. See *Postprocessing for Time-Dependent Problems* (p. 1267) for details.

Important

Note that *mean* statistics are collected only in interior cells and not on wall surfaces. Therefore, when node or cell values of mean quantities are plotted on the wall surface, you are actually plotting values in nearby cells attached to the wall.

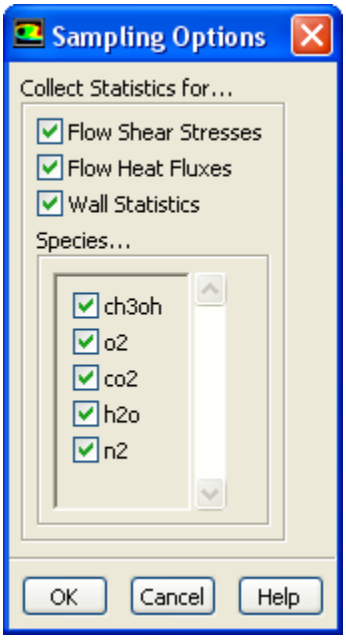
There may be cases when you want to control what set of variables are available for postprocessing. To enable or disable certain variables, go to

Run Calculation → **Sampling Options...**

The **Sampling Options** dialog box will open, where you can enable or disable the statistics shown in *Figure 13.19* (p. 690).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.19 Sampling Options Dialog Box



Important

When including or excluding statistics on variables, it is recommended that you re-initialize the flow statistics.

13.18.3. Troubleshooting

You can use the postprocessing options not only for the purpose of interpreting your results but also for investigating any anomalies that may appear in the solution. For instance, you may want to plot contours of the k field to check if there are any regions where k is erroneously large or small. You should see a high k region in the region where the production of k is large. You may want to display the turbulent viscosity ratio field in order to see whether or not the turbulence takes full effect. Usually the turbulent viscosity is at least two orders of magnitude larger than molecular viscosity for fully-developed turbulent flows modeled using the RANS approach (i.e., not using LES). You may also want to see whether you are using an adequate near-wall mesh for the enhanced wall treatment. To ensure this, you can display filled contours of Re_y (turbulent Reynolds number) overlaid on the mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 14: Modeling Heat Transfer

This chapter provides details about the heat transfer models available in ANSYS FLUENT. Information is presented in the following sections:

14.1. Introduction
14.2. Modeling Conductive and Convective Heat Transfer
14.3. Modeling Radiation
14.4. Modeling Periodic Heat Transfer

14.1. Introduction

The flow of thermal energy from matter occupying one region in space to matter occupying a different region in space is known as heat transfer. Heat transfer can occur by three main methods: conduction, convection, and radiation. Physical models involving conduction and/or convection only are the simplest (*Modeling Conductive and Convective Heat Transfer* (p. 691)), while buoyancy-driven flow or natural convection (*Natural Convection and Buoyancy-Driven Flows* (p. 697)), and radiation models (*Modeling Radiation* (p. 704)) are more complex. Depending on your problem, ANSYS FLUENT will solve a variation of the energy equation that takes into account the heat transfer methods you have specified. ANSYS FLUENT is also able to predict heat transfer in periodically repeating geometries (*Modeling Periodic Heat Transfer* (p. 763)), thus greatly reducing the required computational effort in certain cases.

14.2. Modeling Conductive and Convective Heat Transfer

ANSYS FLUENT allows you to include heat transfer within the fluid and/or solid regions in your model. Problems ranging from thermal mixing within a fluid to conduction in composite solids can thus be handled by ANSYS FLUENT.

When your ANSYS FLUENT model includes heat transfer you will need to activate the relevant physical models, supply thermal boundary conditions, and input material properties that govern heat transfer and/or vary with temperature as part of the setup. For information about heat transfer theory, see Heat Transfer Theory in the Theory Guide. Information about heat transfer theory and how to set up and use heat transfer in your ANSYS FLUENT model is presented in the following subsections:

14.2.1. Solving Heat Transfer Problems
14.2.2. Solution Strategies for Heat Transfer Modeling
14.2.3. Postprocessing Heat Transfer Quantities
14.2.4. Natural Convection and Buoyancy-Driven Flows
14.2.5. Shell Conduction Considerations

14.2.1. Solving Heat Transfer Problems

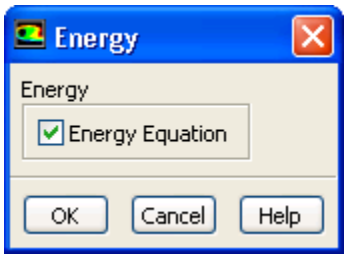
The procedure for setting up a heat transfer problem is described below. Note that this procedure includes only those steps necessary for the heat transfer model itself; you will need to set up other models, boundary conditions, etc. as usual.

1. To activate the calculation of heat transfer, enable the **Energy Equation** option in the **Energy** dialog box (*Figure 14.1* (p. 692)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Models → **Energy** → **Edit...**

Figure 14.1 The Energy Dialog Box



2. (Optional, pressure-based solver only.) If you are modeling viscous flow and you want to include the viscous heating terms in the energy equation, enable the **Viscous Heating** option in the **Viscous Model** dialog box.

   Models → **Viscous** → **Edit...**

   As noted in Inclusion of the Viscous Dissipation Terms in the Theory Guide, the viscous heating terms in the energy equation are (by default) ignored by ANSYS FLUENT when the pressure-based solver is used. They are always included for the density-based solver. Viscous dissipation should be enabled when the shear stress in the fluid is large (e.g., in lubrication problems) and/or in high-velocity, compressible flows. See Equation 5–9 in the Theory Guide.

3. Define thermal boundary conditions at flow inlets, flow outlets, and walls.

   Boundary Conditions

   At flow inlets and exits you will set the temperature; at walls you may use any of the following thermal conditions:

   - specified heat flux
   - specified temperature
   - convective heat transfer
   - external radiation
   - combined external radiation and external convective heat transfer

   Thermal Boundary Conditions at Walls (p. 309) provides details on the model inputs that govern these thermal boundary conditions. The default thermal boundary condition at inlets is a specified temperature of 300 K; at walls the default condition is zero heat flux (adiabatic). See *Cell Zone and Boundary Conditions* (p. 205) for details about boundary condition inputs.

   Important

   If your heat transfer application involves two separated fluid regions, see the information provided below.

4. Define material properties for heat transfer.

   Materials

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat capacity and thermal conductivity must be defined, and you can specify many properties as functions of temperature as described in *Physical Properties* (p. 385).

14.2.1.1. Limiting the Predicted Temperature Range

For stability reasons, ANSYS FLUENT includes a limit on the predicted temperature range. The purpose of the temperature ceiling and floor is to improve the stability of calculations in which the temperature should physically lie within known limits. Sometimes intermediate solutions of the equations give rise to temperatures beyond these limits for which property definitions, etc. are not well defined.

The temperature limits keep the temperatures within the expected range for your problem. If the ANSYS FLUENT calculation predicts a temperature above the maximum limit, the stored temperature values are "pegged" at this maximum value. The default for the temperature ceiling is 5000 K. If the ANSYS FLUENT calculation predicts a temperature below the minimum limit, the stored temperature values are "pegged" at this minimum value. The default for the temperature minimum is 1 K.

If you expect the temperature in your domain to exceed 5000 K, use the **Solution Limits** dialog box to increase the **Maximum Temperature**.

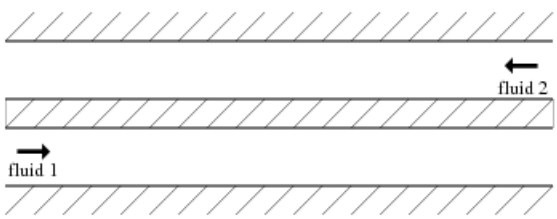
Solution Controls → **Limits...**

14.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions

If your heat transfer application involves two fluid regions separated by a solid zone or a wall, as illustrated in *Figure 14.2* (p. 693), you will need to define the problem with some care. Specifically:

- You should not use outflow boundary conditions in either fluid.
- You can establish separate fluid properties by selecting a different fluid material for each zone. For species calculations, however, you can only select a single mixture material for the entire domain.

Figure 14.2 Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams



14.2.2. Solution Strategies for Heat Transfer Modeling

Although many simple heat transfer problems can be successfully solved using the default solution parameters assumed by ANSYS FLUENT, you may accelerate the convergence of your problem and/or improve the stability of the solution process using some of the guidelines provided in this section.

14.2.2.1. Under-Relaxation of the Energy Equation

When you use the pressure-based solver, ANSYS FLUENT under-relaxes the energy equation using the under-relaxation parameter defined by you in the **Solution Controls** task page, as described in *Setting Under-Relaxation Factors* (p. 1217).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Controls

If you are using the non-adiabatic non-premixed combustion model, you will set the energy under-relaxation factor as usual but you will also set an under-relaxation factor for temperature, as described below.

ANSYS FLUENT uses a default under-relaxation factor of 1.0 for the energy equation, regardless of the form in which it is solved (temperature or enthalpy). In problems where the energy field impacts the fluid flow (via temperature-dependent properties or buoyancy) you should use a lower value for the under-relaxation factor, in the range of 0.8–1.0. In problems where the flow field is decoupled from the temperature field (no temperature-dependent properties or buoyancy forces), you can usually retain the default value of 1.0.

14.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved

When the enthalpy form of the energy equation is solved (i.e., when you are using the non-adiabatic non-premixed combustion model), ANSYS FLUENT also under-relaxes the temperature, updating the temperature by only a fraction of the change that would result from the change in the (under-relaxed) enthalpy values. This second level of under-relaxation can be used to good advantage when you would like to let the enthalpy field change rapidly, but the temperature response (and its effect on fluid properties) to lag. ANSYS FLUENT uses a default setting of 1.0 for the under-relaxation on temperature and you can modify this setting using the **Solution Controls** task page.

14.2.2.3. Disabling the Species Diffusion Term

If you are solving for species transport using the pressure-based solver and you encounter convergence difficulties, you may want to consider disabling the **Diffusion Energy Source** option in the **Species Model** dialog box.

Models → **Species** → **Edit...**

When this option is disabled, ANSYS FLUENT will neglect the effects of species diffusion on the energy equation.

Note that species diffusion effects are always included when the density-based solver is used.

14.2.2.4. Step-by-Step Solutions

Often the most efficient strategy for predicting heat transfer is to compute an isothermal flow first and then add the calculation of the energy equation. The procedure differs slightly, depending on whether or not the flow and heat transfer are coupled.

14.2.2.4.1. Decoupled Flow and Heat Transfer Calculations

If your flow and heat transfer are decoupled (no temperature-dependent properties or buoyancy forces), you can first solve the isothermal flow (energy equation turned off) to yield a converged flow-field solution and then solve the energy transport equation alone.

Important

Since the density-based solver always solves the flow and energy equations together, the procedure for solving for energy alone applies to the pressure-based solver, only.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can temporarily disable the flow equations or the energy equation by disabling the **Energy** option in the **Equations** dialog box

Solution Controls → **Equations...**

or

Models → **Energy** → **Edit...**

14.2.2.4.2. Coupled Flow and Heat Transfer Calculations

If the flow and heat transfer are coupled (i.e., your model includes temperature-dependent properties or buoyancy forces), you can first solve the flow equations before enabling energy. Once you have a converged flow-field solution, you can enable energy and solve the flow and energy equations simultaneously to complete the heat transfer simulation.

14.2.3. Postprocessing Heat Transfer Quantities

14.2.3.1. Available Variables for Postprocessing

ANSYS FLUENT provides reporting options for simulations involving heat transfer. You can generate graphical plots or reports of the following variables/functions:

- **Static Temperature**
- **Total Temperature**
- **Enthalpy**
- **Relative Total Temperature**
- **Rothalpy**
- **Wall Temperature (Outer Surface)**
- **Wall Temperature (Inner Surface)**
- **Total Enthalpy**
- **Total Enthalpy Deviation**
- **Entropy**
- **Total Energy**
- **Internal Energy**
- **Total Surface Heat Flux**
- **Surface Heat Transfer Coef.**
- **Surface Nusselt Number**
- **Surface Stanton Number**

The first 12 variables listed above are contained in the **Temperature...** category of the variable selection drop-down list that appears in postprocessing dialog boxes, and the remaining variables are in the **Wall Fluxes...** category. See *Field Function Definitions* (p. 1529) for their definitions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.3.2. Definition of Enthalpy and Energy in Reports and Displays

The definitions of the reported values of enthalpy and energy will be different depending on whether the flow is compressible or incompressible.

14.2.3.3. Reporting Heat Transfer Through Boundaries

You can use the **Flux Reports** dialog box to compute the heat transfer through each boundary of the domain, or to sum the heat transfer through all boundaries to check the heat balance.

Reports → **Fluxes** → **Set Up...**

It is recommended that you perform a heat balance check to ensure that your solution is truly converged.

14.2.3.4. Reporting Heat Transfer Through a Surface

You can use the **Surface Integrals** dialog box to compute the heat transfer through any boundary or any surface created using the methods described in *Creating Surfaces for Displaying and Reporting Data* (p. 1355).

Reports → **Surface Integrals** → **Set Up...**

To report the mass flow rate of enthalpy

$$Q = \int H \rho \vec{v} \cdot d\vec{A} \qquad \textbf{(14–1)}$$

choose **Flow Rate** for the **Report Type** in the **Surface Integrals** dialog box, select **Enthalpy** (in the **Temperature...** category) as the **Field Variable**, and select the surface(s) on which to integrate.

14.2.3.5. Reporting Averaged Heat Transfer Coefficients

The **Surface Integrals** dialog box can also be used to generate a report of averaged heat transfer coefficient h on a surface ($\frac{1}{A}\int h dA$).

Reports → **Surface Integrals** → **Set Up...**

In the **Surface Integrals** dialog box, choose **Area-Weighted Average** for **Report Type**, select **Surface Heat Transfer Coef.** (in the **Wall Fluxes...** category) as the **Field Variable**, and select the surface.

14.2.3.6. Exporting Heat Flux Data

It is possible to export heat flux data on wall zones (including radiation) to a generic file that you can examine or use in an external program. To save a heat flux file, you will use the `custom-heat-flux` text command.

`file` → `export` → `custom-heat-flux`

Heat transfer data will be exported in the following free format for each face zone that you select for export:

```
zone-name nfaces
x_f y_f z_f A Q T_w T_c HTC
.
.
.
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Each block of data starts with the name of the face zone (`zone-name`) and the number of faces in the zone (`nfaces`). Next there is a line for each face (i.e., `nfaces` lines), each containing the components of the face centroid (`x_f`, `y_f`, and, in 3D, `z_f`), the face area (A), the heat transfer rate (Q), the face temperature (`T_w`), the adjacent cell temperature (`T_c`), and the heat transfer coefficient (HTC). If the heat transfer coefficient is calculated based on wall function (*Equation 33–44* (p. 1573)), then Q is the convective heat transfer rate. Otherwise, Q will be the total heat transfer rate, including radiation heat transfer.

14.2.4. Natural Convection and Buoyancy-Driven Flows

When heat is added to a fluid and the fluid density varies with temperature, a flow can be induced due to the force of gravity acting on the density variations. Such buoyancy-driven flows are termed natural-convection (or mixed-convection) flows and can be modeled by ANSYS FLUENT.

For more information about the theory behind natural convection and buoyancy-driven flows, see Natural Convection and Buoyancy-Driven Flows Theory in the Theory Guide.

14.2.4.1. Modeling Natural Convection in a Closed Domain

When you model natural convection inside a closed domain, the solution will depend on the mass inside the domain. Since this mass will not be known unless the density is known, you must model the flow in one of the following ways:

- Perform a transient calculation. In this approach, the initial density will be computed from the initial pressure and temperature, so the initial mass is known. As the solution progresses over time, this mass will be properly conserved. If the temperature differences in your domain are large, you must follow this approach.
- Perform a steady-state calculation using the Boussinesq model (described in *The Boussinesq Model* (p. 697)). In this approach, you will specify a constant density, so the mass is properly specified. This approach is valid only if the temperature differences in the domain are small. If not, you use the transient approach.

Important

For a closed domain, you can use the *incompressible* ideal gas law only with a *fixed* operating pressure. It *cannot* be used with a floating operating pressure. You can use the *compressible* ideal gas law with either *floating* or *fixed* operating pressure.

See *Floating Operating Pressure* (p. 507) for more information about the floating operating pressure option.

14.2.4.2. The Boussinesq Model

For many natural-convection flows, you can get faster convergence with the Boussinesq model than you can get by setting up the problem with fluid density as a function of temperature. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

$$\left(\rho-\rho_0\right)g \approx -\rho_0\beta\left(T-T_0\right)g \qquad \textbf{(14–2)}$$

where ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. *Equation 14–2* (p. 697) is obtained by using the Boussinesq approximation $\rho=\rho_0\left(1-\beta\Delta T\right)$ to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid when $\beta(T - T_0) \ll 1$.

14.2.4.3. Limitations of the Boussinesq Model

The Boussinesq model should not be used if the temperature differences in the domain are large. In addition, it cannot be used with species calculations, combustion, or reacting flows.

14.2.4.4. Steps in Solving Buoyancy-Driven Flow Problems

The procedure for including buoyancy forces in the simulation of mixed or natural convection flows is described below.

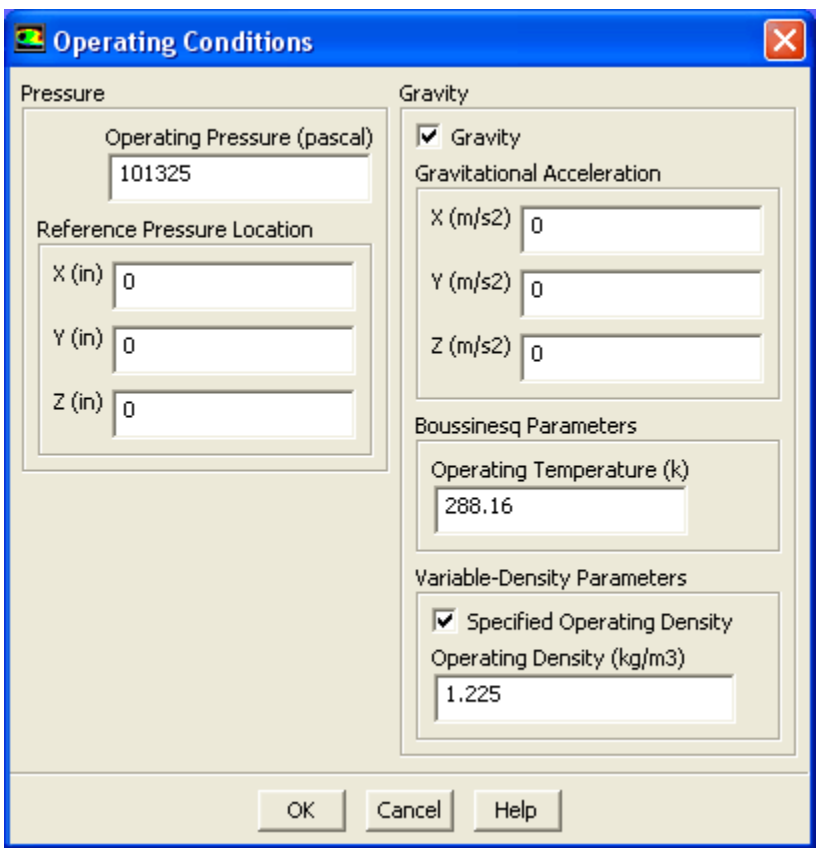
1. Activate the calculation of heat transfer, by enabling the **Energy** option in the **Energy** dialog box.

   Models → **Energy** → **Edit...**

2. Define the operating conditions in the **Operating Conditions** dialog box (*Figure 14.3* (p. 698)).

   Cell Zone Conditions → **Operating Conditions...**

   Figure 14.3 The Operating Conditions Dialog Box



   a. Enable the **Gravity** option under **Gravity**.

   b. Enter the appropriate values in the **X**, **Y**, and (for 3D) **Z** fields for **Gravitational Acceleration** for each Cartesian coordinate direction. (Note that the default gravitational acceleration in ANSYS FLUENT is zero.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. If you are using the incompressible ideal gas law, check that the **Operating Pressure** is set to an appropriate (non-zero) value.

d. Depending on whether or not you use the Boussinesq approximation, specify the appropriate parameters described below:

- If you are not using the Boussinesq model, the inputs are as follows:

  i. If necessary, enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and enter a value for the **Operating Density**. See below for details.

  ii. Define the fluid density as a function of temperature as described in *Defining Properties Using Temperature-Dependent Functions* (p. 398) and *Density* (p. 403).

  Materials

- If you are using the Boussinesq model (described in *The Boussinesq Model* (p. 697)) the inputs are as follows:

  i. Enter the **Operating Temperature** (T_0 in *Equation 14–2* (p. 697)) in the **Operating Conditions** dialog box.

  ii. Select **boussinesq** in the drop-down list for **Density** in the **Create/Edit Materials** dialog box as described in *Defining Properties Using Temperature-Dependent Functions* (p. 398) and *Density* (p. 403), and enter a constant value.

  iii. Also in the **Create/Edit Materials** dialog box, enter an appropriate value for the **Thermal Expansion Coefficient** (β in *Equation 14–2* (p. 697)) for the fluid material.

Note that if your model involves multiple fluid materials you can choose whether or not to use the Boussinesq model for each material. As a result, you may have some materials using the Boussinesq model and others not. In such cases, you will need to set all the parameters described above in this step.

3. Define the boundary conditions.

Boundary Conditions

The boundary pressures that you input at pressure inlet and outlet boundaries are the redefined pressures as given by *Equation 14–3* (p. 700). In general you should enter equal pressures, p', at the inlet and exit boundaries of your ANSYS FLUENT model if there are no externally-imposed pressure gradients.

4. Set the parameters that control the solution.

Solution Methods

a. Select **Body Force Weighted** or **Second Order** in the drop-down list for **Pressure** under **Spatial Discretization** in the **Solution Methods** task page.

b. If you are using the pressure-based solver, selecting **PRESTO!** as the **Spatial Discretization** method for **Pressure** is the recommended approach.

c. Add cells near the walls to resolve boundary layers, if necessary.

See also *Solving Heat Transfer Problems* (p. 691) for information on setting up heat transfer calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.4.5. Operating Density

When the Boussinesq approximation is not used, the operating density ρ_0 appears in the body-force term in the momentum equations as $\left(\rho-\rho_0\right)g$. This form of the body-force term follows from the redefinition of pressure in ANSYS FLUENT as

$$p_s'=p_s-\rho_0 g x \tag{14–3}$$

The hydrostatic pressure in a fluid at rest is then

$$p_s'=0 \tag{14–4}$$

14.2.4.5.1. Setting the Operating Density

By default, ANSYS FLUENT will compute the operating density by averaging over all cells. In some cases, you may obtain better results if you explicitly specify the operating density instead of having the solver compute it for you. For example, if you are solving a natural-convection problem with a pressure boundary, it is important to understand that the pressure you are specifying is p_s' in *Equation 14–3* (p. 700). Although you will know the actual pressure p_s, you will need to know the operating density ρ_0 in order to determine p_s' from p_s. Therefore, you should explicitly specify the operating density rather than use the computed average. The specified value should, however, be representative of the average value.

In some cases the specification of an operating density will improve convergence behavior, rather than the actual results. For such cases use the approximate bulk density value as the operating density and be sure that the value you choose is appropriate for the characteristic temperature in the domain.

Note that if you are using the Boussinesq approximation for all fluid materials, the operating density ρ_0 does not appear in the body-force term of the momentum equation. Consequently, you need not specify it.

14.2.4.6. Solution Strategies for Buoyancy-Driven Flows

For high-Rayleigh-number flows you may want to consider the solution guidelines below. In addition, the guidelines presented in *Solution Strategies for Heat Transfer Modeling* (p. 693) for solving other heat transfer problems can also be applied to buoyancy-driven flows. No steady-state solution exists for some laminar, high-Rayleigh-number flows.

14.2.4.6.1. Guidelines for Solving High-Rayleigh-Number Flows

When you are solving a high-Rayleigh-number flow ($Ra>10^8$), you should follow one of the procedures outlined below for best results.

The first procedure uses a steady-state approach:

1. Start the solution with a lower value of Rayleigh number (e.g., 10^7) and run it to convergence using the first-order scheme.
2. To change the effective Rayleigh number, change the value of gravitational acceleration (e.g., from 9.8 to 0.098 to reduce the Rayleigh number by two orders of magnitude).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use the resulting data file as an initial guess for the higher Rayleigh number and start the higher-Rayleigh-number solution using the first-order scheme.
4. After you obtain a solution with the first-order scheme you may continue the calculation with a higher-order scheme.

The second procedure uses a time-dependent approach to obtain a steady-state solution [31] (p. 2210):

1. Start the solution from a steady-state solution obtained for the same or a lower Rayleigh number.
2. Estimate the time constant as [10] (p. 2209)

$$\tau = \frac{L}{U} \sim \frac{L^2}{\alpha}(PrRa)^{-1/2} = \frac{L}{\sqrt{g\beta\Delta TL}} \qquad \textbf{(14–5)}$$

where L and U are the length and velocity scales, respectively. Use a time step Δt such that

$$\Delta t \approx \frac{\tau}{4} \qquad \textbf{(14–6)}$$

Using a larger time step Δt may lead to divergence.

3. After oscillations with a typical frequency of $f\tau =$ 0.05–0.09 have decayed, the solution reaches steady state. Note that τ is the time constant estimated in *Equation 14–5* (p. 701) and f is the oscillation frequency in Hz. In general this solution process may take as many as 5000 time steps to reach steady state.

14.2.4.7. Postprocessing Buoyancy-Driven Flows

The postprocessing reports of interest for buoyancy-driven flows are the same as for other heat transfer calculations. See *Postprocessing Heat Transfer Quantities* (p. 695) for details.

14.2.5. Shell Conduction Considerations

14.2.5.1. Introduction

By default, ANSYS FLUENT treats walls as zero thickness presenting no thermal resistance to heat transfer across them. If a thickness is specified for walls then the appropriate thermal resistance across the wall thickness is imposed, although conduction is considered in the wall in the normal direction only. There are applications, however, where conduction in the planar direction of the wall is also important. For these applications, you have two options: you can either mesh the thickness or you can use the shell conduction approach. Shell conduction can be used to model thin sheets without the need to mesh the wall thickness in a preprocessor. When the shell conduction approach is utilized, you have the ability to easily switch on and off conjugate heat transfer on any wall. When you specify a thickness for the wall, a material property, and enable **Shell Conduction** in the **Wall** dialog box (as described in *Shell Conduction in Thin-Walls* (p. 313)) or in the **Shell Conduction Walls** dialog box (*Managing Shell Conduction Walls* (p. 703)), then during the solution process ANSYS FLUENT automatically grows a layer of prism cells or hex cells for the wall, depending on the type of face mesh that is utilized.

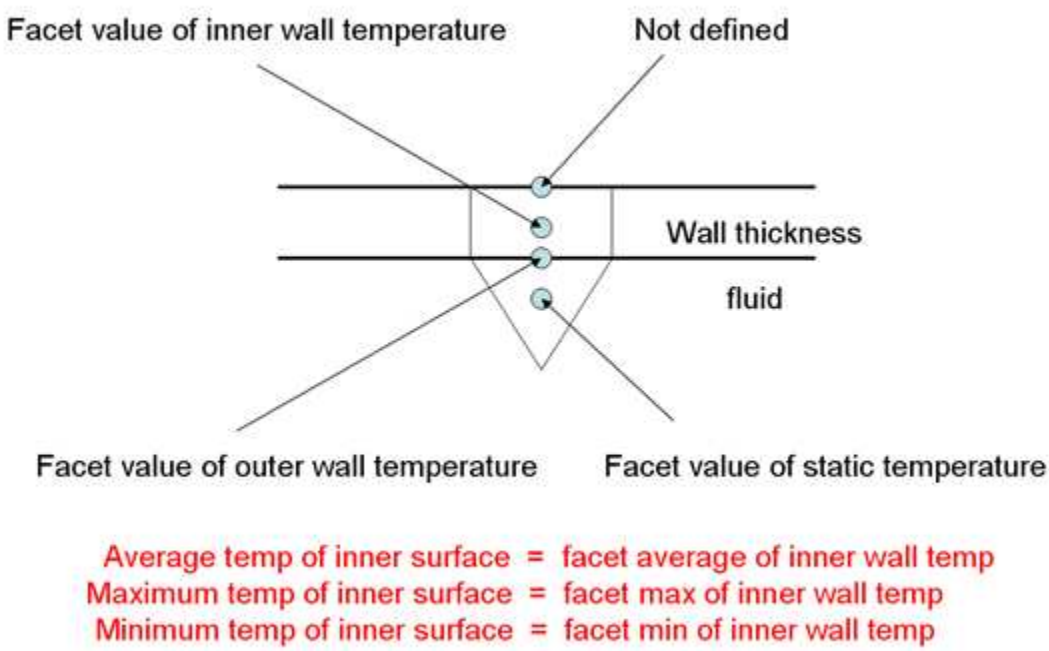
Shell conduction can be used to account for thermal mass in transient thermal analysis problems such as soaking. It can also be used for multiple junctions and allows heat conduction through the junctions. Shell conduction can be applied on boundary walls as well as internal walls.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.5.2. Physical Treatment

In the case of shell conduction that is applied on a boundary wall, the boundary condition that you specify on the original wall is applied to the inner surface and the original wall is treated as a coupled wall (*Figure 14.4* (p. 702)). Note however, that internal emissivity is applied at the outer surface. The shell boundaries (the sides of the shell zone) need boundary conditions as well. If the wall with shell conduction is connected to another wall that has no shell conduction, the shell side will take its boundary condition. The sides will be adiabatic if they are connected to face zones having a boundary condition type other than a "wall". If the attached wall has shell conduction, the common sides at the junction will be coupled.

Figure 14.4 A Case for Shell Conduction



14.2.5.3. Limitations of Shell Conduction Walls

The following is a list of limitations for the shell conduction model:

- Shells cannot be created on non-conformal interfaces.
- Shell conduction cannot be used on moving wall zones.
- Shell conduction cannot be used with FMG initialization.
- Shell conduction is not available for 2D.
- Shell conduction is available only when the pressure-based solver is used.
- Shell conduction cannot be used with the non-premixed or partially-premixed combustion model.
- When used in conjunction with the Discrete Ordinates (DO) radiation model, shell conducting walls cannot be semi-transparent and should not be used with the DO/Energy Coupling Method.
- Shell conducting walls cannot be split or merged. If you need to split or merge a shell conducting wall, you will need to turn off the **Shell Conduction** option for the wall (in the **Wall** dialog box, perform the split or merge operation, and then enable **Shell Conduction** for the new wall zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The shell conduction model cannot be used on a wall zone that has been adapted. If you want to perform adaption elsewhere in the computational domain, be sure to use the mask register described in *Manipulating Adaption Registers* (p. 1343). This will ensure that adaption is not performed on the shell conducting wall.
- Fluxes at the ends of a shell conducting wall are not included in heat balance reports. These fluxes are accounted for correctly in the ANSYS FLUENT solution, but are not listed in the flux report.

14.2.5.4. Managing Shell Conduction Walls

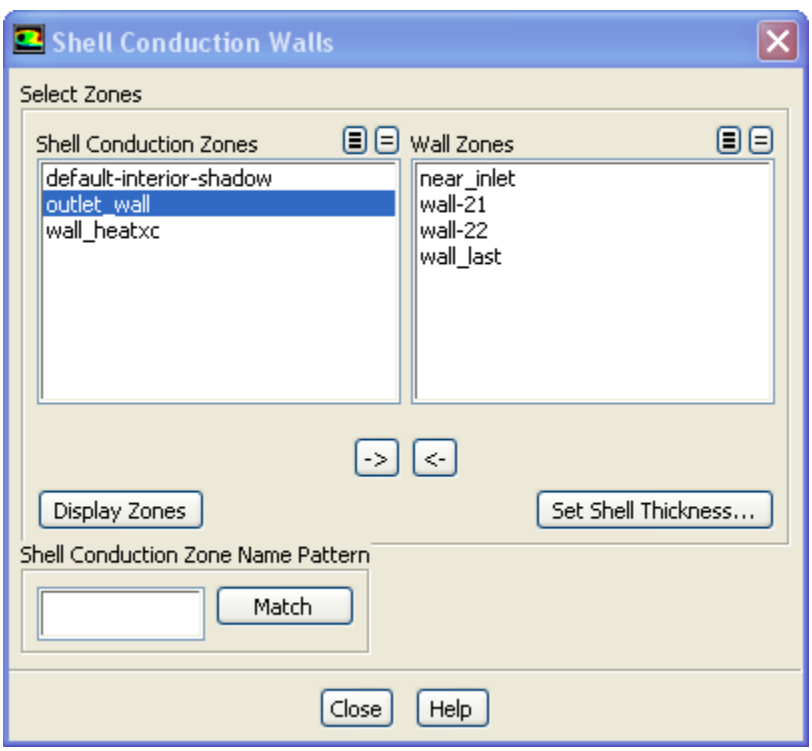
The **Shell Conduction Walls** dialog box is a convenient application, which allows you to manage, define, and display shell conduction zones all in one location. You can

- Display any wall zone.
- Enable or disable shell conduction for a wall.
- Set the thickness for shell conduction walls.

Note that you can still enable shell conduction and set the thickness of a wall in the **Wall** boundary conditions dialog box, as described in *Shell Conduction in Thin-Walls* (p. 313). However, the **Shell Conduction Walls** dialog box provides you with an alternative to do such activities from a single dialog box for all the walls, instead of visiting the individual **Wall** boundary condition dialog boxes.

Define → Shell Conduction Walls...

Figure 14.5 The Shell Conduction Walls Dialog Box



To manage shell conduction using the **Shell Conduction Walls** dialog box, do the following:

1. Use the − > and < − arrows to enable or disable shell conduction. Zones with shell conduction enabled are listed under **Shell Conduction Zones** and those without shell conduction are listed under **Wall**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zones. In essence, the − > button disables shell conduction and the < − button enables shell conduction.

2. Click the **Display Zones** button to display the selected wall(s) in the graphics window. Note that you can select walls with or without shell conduction. The zones will be displayed with different colors depending on the option selected in **Mesh Colors** dialog box accessible from the **Mesh Display** dialog box.
3. Click the **Set Shell Thickness...** button to open the **Set Shell Thickness** dialog box (*Figure 14.6* (p. 704)), where you can set the thicknesses of the selected **Shell Conduction Zones**.

Figure 14.6 The Set Shell Thickness Dialog Box

   a. If you want to apply the same thickness to all the selected shell conduction zones, enter the value in the **Thickness** field and click the **Apply** button.
   b. If you want to apply different thicknesses to the selected shell conduction zones, enter the values against each zone listed in the **Shell Conduction Zones** group box and click the **OK** button.
4. You can use the **Shell Conduction Zone Name Pattern** to select only shell conduction zones with names that match the specified pattern.

14.2.5.5. Initialization

Shell zones can be patched using the **Patch** dialog box.

Solution Initialization → **Patch...**

14.2.5.6. Postprocessing

Shell zones can be postprocessed. The shell cell temperature is stored in the **Temperature** variable (inner surface). If a more detailed analysis of the solid zone and surfaces is required, then you should consider using a layer of solid zones in your model.

14.3. Modeling Radiation

Information about radiation modeling is presented in the following sections:

14.3.1. Using the Radiation Models
14.3.2. Setting Up the P-1 Model with Non-Gray Radiation
14.3.3. Setting Up the DTRM
14.3.4. Setting Up the S2S Model
14.3.5. Setting Up the DO Model

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.6. Defining Material Properties for Radiation
14.3.7. Defining Boundary Conditions for Radiation
14.3.8. Solution Strategies for Radiation Modeling
14.3.9. Postprocessing Radiation Quantities
14.3.10. Solar Load Model

For theoretical information about the radiation models in ANSYS FLUENT, refer to Modeling Radiation in the Theory Guide.

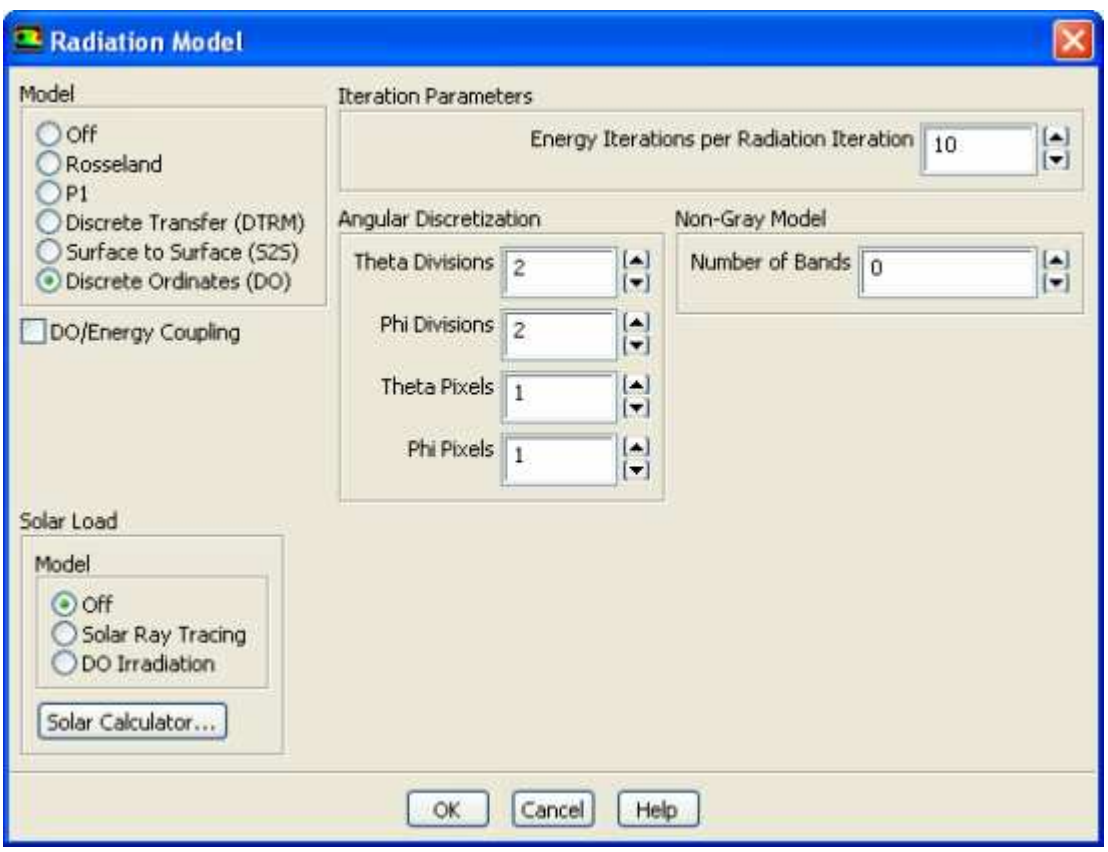
14.3.1. Using the Radiation Models

The procedure for setting up and solving a radiation problem is outlined below, and described in detail in referenced sections. Steps that are relevant only for a particular radiation model are noted as such. The steps that are pertinent to radiation modeling, are shown here. For information about inputs related to other models that you are using in conjunction with radiation, see the appropriate sections for those models.

1. Activate radiative heat transfer by selecting a radiation model (**Rosseland**, **P1**, **Discrete Transfer (DTRM)**, **Surface to Surface (S2S)**, or **Discrete Ordinates**) under **Model** in the **Radiation Model** dialog box (*Figure 14.7* (p. 705)).

   Models → **Radiation** → **Edit...**

   Figure 14.7 The Radiation Model Dialog Box (DO Model)



Important

The **Rosseland** model can be used only with the pressure-based solver.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that when the P-1, the DTRM, the S2S, or the DO model is activated, the **Radiation Model** dialog box expands to show additional parameters. These parameters will not appear if you select the Rosseland model. If you are running a 3d case, you will have the added option of using the solar load model. The solar load options will be displayed in the dialog box, below the radiation model settings.

When the radiation model is active, the radiation fluxes will be included in the solution of the energy equation at each iteration. If you set up a problem with the radiation model turned on, and you then decide to turn it off completely, you must select the **Off** button in the **Radiation Model** dialog box.

Note that when you enable a radiation model, ANSYS FLUENT will automatically enable the energy equation so that step is not needed.

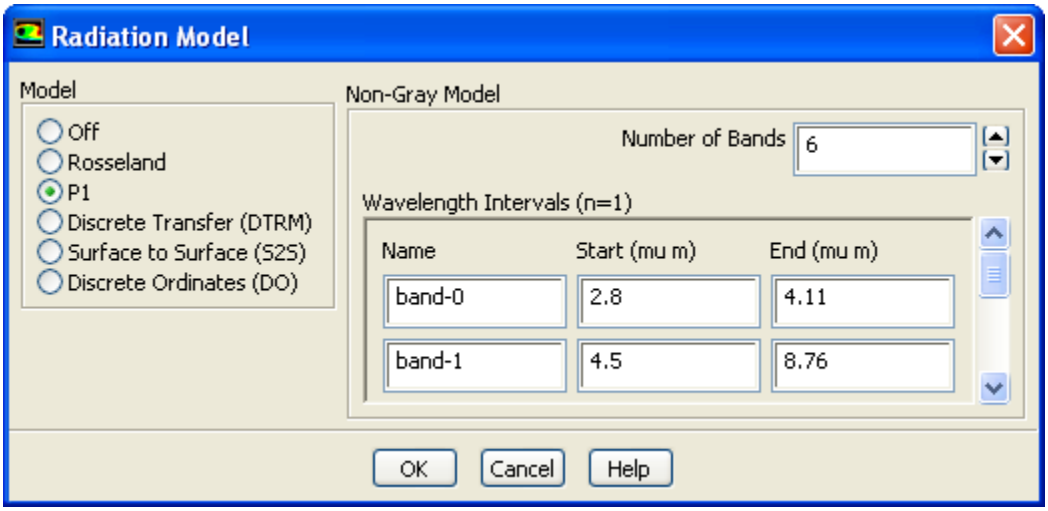
2. Set the appropriate radiation parameters.
   a. If you are modeling non-gray radiation using the P-1 model, define the non-gray radiation parameters as described in *Setting Up the P-1 Model with Non-Gray Radiation* (p. 706).
   b. If you are using the DTRM, define the ray tracing as described in *Setting Up the DTRM* (p. 707).
   c. If you are using the S2S model, define the surface clusters and view factors settings and compute or read the view factors as described in *Setting Up the S2S Model* (p. 709).
   d. If you are using the DO model, choose **DO/Energy Coupling** if desired, define the angular discretization as described in *Setting Up the DO Model* (p. 720) and, if relevant, define the non-gray radiation parameters as described in *Defining Non-Gray Radiation for the DO Model* (p. 721).
3. Define the material properties as described in *Defining Material Properties for Radiation* (p. 723).
4. Define the boundary conditions as described in *Defining Boundary Conditions for Radiation* (p. 724). If your model contains a semi-transparent medium, see the information below on setting up semi-transparent media.
5. Set the parameters that control the solution (DTRM, DO, S2S, and P-1 only) as described in *Solution Strategies for Radiation Modeling* (p. 732).
6. Run the solution as described in *Running the Calculation* (p. 735).
7. Postprocess the results as described in *Postprocessing Radiation Quantities* (p. 736).

14.3.2. Setting Up the P-1 Model with Non-Gray Radiation

If you want to model non-gray radiation using the P-1 model, you can specify the **Number of Bands** under **Non-Gray Model** in the expanded **Radiation Model** dialog box (*Figure 14.8* (p. 707)). By default, the **Number of Bands** is set to zero, indicating that only gray radiation will be modeled. Because the cost of computation increases directly with the number of bands, you should try to minimize the number of bands used. When a non-zero **Number of Bands** is specified, the **Radiation Model** dialog box will expand once again to show the **Wavelength Intervals**. For each wavelength band, you can specify a **Name**, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum ($n = 1$). For more information about non-gray radiation calculations, see *Defining Non-Gray Radiation for the DO Model* (p. 721).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.8 The Radiation Model Dialog Box (Non-Gray P-1 Model)



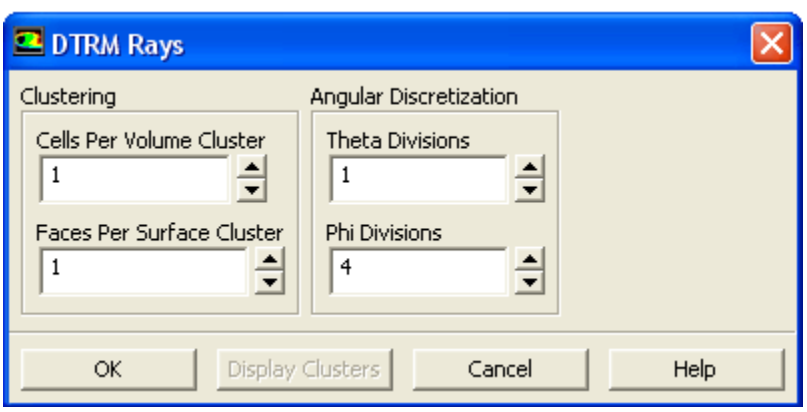
ANSYS FLUENT allows you to use a user-defined function (UDF) to modify the emissivity weighting factor $F(0 \rightarrow n\lambda_2 T) - F(0 \rightarrow n\lambda_1 T)$ (which otherwise defaults to the black body emission factor obtained from a standard Planck distribution). The emissivity weighting factor appears in the emission term of the radiative transfer equation for the non-gray model, as shown in Equation 5–25 in the Theory Guide. For more information, see the UDF Manual.

14.3.3. Setting Up the DTRM

14.3.3.1. Defining the Rays

When you select the **Discrete Transfer** model and click **OK** in the **Radiation Model** dialog box, the **DTRM Rays** dialog box (*Figure 14.9* (p. 707)) will open automatically. If you need to modify the current settings later in the problem setup or solution procedure, you can open this dialog box manually using the **Define/DTRM Rays...** menu item.

Figure 14.9 The DTRM Rays Dialog Box



In this dialog box you will set parameters for and create the rays and clusters discussed in The DTRM Equations in the Theory Guide.

The procedure is as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To control the number of radiating surfaces and absorbing cells, set the **Cells Per Volume Cluster** and **Faces Per Surface Cluster**. (See the explanation below.)
2. To control the number of rays being traced, set the number of **Theta Divisions** and **Phi Divisions**. (Guidelines are provided below.)
3. When you click **OK** in the **DTRM Rays** dialog box, *The **Select File** Dialog Box* (p. 33) will open prompting you for the name of the "ray file". After you have specified the file name and chosen whether to write a binary ray file, ANSYS FLUENT will write the ray file and then read it afterward. During the write process the status of the DTRM ray tracing will be reported in the ANSYS FLUENT console. For example:

   ```
   Completed 25% tracing of DTRM rays

   Completed 50% tracing of DTRM rays

   Completed 75% tracing of DTRM rays

   Completed 100 % tracing of DTRM rays
   ```

   See following sections for details on **DTRM Rays** dialog box inputs.

Important

If you cancel the **DTRM Rays** dialog box without writing and reading the ray file, the DTRM will be disabled.

14.3.3.2. Controlling the Clusters

Your inputs for **Cells Per Volume Cluster** and **Faces Per Surface Cluster** will control the number of radiating surfaces and absorbing cells. By default, each is set to 1, so the number of surface clusters (radiating surfaces) will be the number of boundary faces, and the number of volume clusters (absorbing cells) will be the number of cells in the domain. For small 2D problems, these are acceptable numbers, but for larger problems you will want to reduce the number of surface and/or volume clusters in order to reduce the ray-tracing expense. See Clustering in the Theory Guide for details about clustering.

14.3.3.3. Controlling the Rays

Your inputs for **Theta Divisions** and **Phi Divisions** will control the number of rays being traced from each surface cluster (radiating surface).

Theta Divisions defines the number of discrete divisions in the angle θ used to define the solid angle about a point P on a surface. The solid angle is defined as θ varies from 0 to 90 degrees (in the Theory Guide), and the default setting of 1 for the number of discrete settings implies that there will be one ray traced from the surface.

Phi Divisions defines the number of discrete divisions in the angle ϕ used to define the solid angle about a point P on a surface. The solid angle is defined as ϕ varies from 0 to 360 degrees (Figure 5.2: "Angles θ and ϕ Defining the Hemispherical Solid Angle About a Point P" in the Theory Guide). The default setting of 4 implies that each ray traced from the surface will be located at a 90° angle, and in combination with the default setting for **Theta Divisions**, above, implies that 4 rays will be traced from each surface control volume. In many cases, it is recommended that you at least double the number of divisions in θ and ϕ.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.3.4. Writing and Reading the DTRM Ray File

After you have activated the DTRM and defined all of the parameters controlling the ray tracing, you must create a ray file which will be read back in and used during the radiation calculation. The ray file contains a description of the ray traces (path lengths, cells traversed by each ray, etc.). This information is stored in the ray file, instead of being recomputed, in order to speed up the calculation process.

By default, a binary ray file will be written. You can also create text (formatted) ray files by turning off the **Write Binary Files** option in *The **Select File** Dialog Box* (p. 33).

Important

Do not write or read a compressed ray file, because ANSYS FLUENT will not be able to access the ray tracing information properly from a compressed ray file.

The ray filename must be specified to ANSYS FLUENT only once. Thereafter, the filename is stored in your case file and the ray file will be automatically read into ANSYS FLUENT whenever the case file is read. ANSYS FLUENT will remind you that it is reading the ray file after it finishes reading the rest of the case file by reporting its progress in the console.

Note that the ray filename stored in your case file may not contain the full name of the directory in which the ray file exists. The full directory name will be stored in the case file only if you initially read the ray file through the GUI (or if you typed in the directory name along with the filename when using the text interface). In the event that the full directory name is absent, the automatic reading of the ray file may fail (since ANSYS FLUENT does not know in which directory to look for the file), and you will need to manually specify the ray file, using the **File/Read/DTRM Rays...** menu item. The safest approaches are to use the GUI when you first read the ray file or to supply the full directory name when using the text interface.

Important

You should recreate the ray file whenever you do anything that changes the mesh, such as:

- change the type of a boundary zone
- adapt or reorder the mesh
- scale the mesh

You can open the **DTRM Rays** dialog box directly with the **Define/DTRM Rays...** menu item.

14.3.3.5. Displaying the Clusters

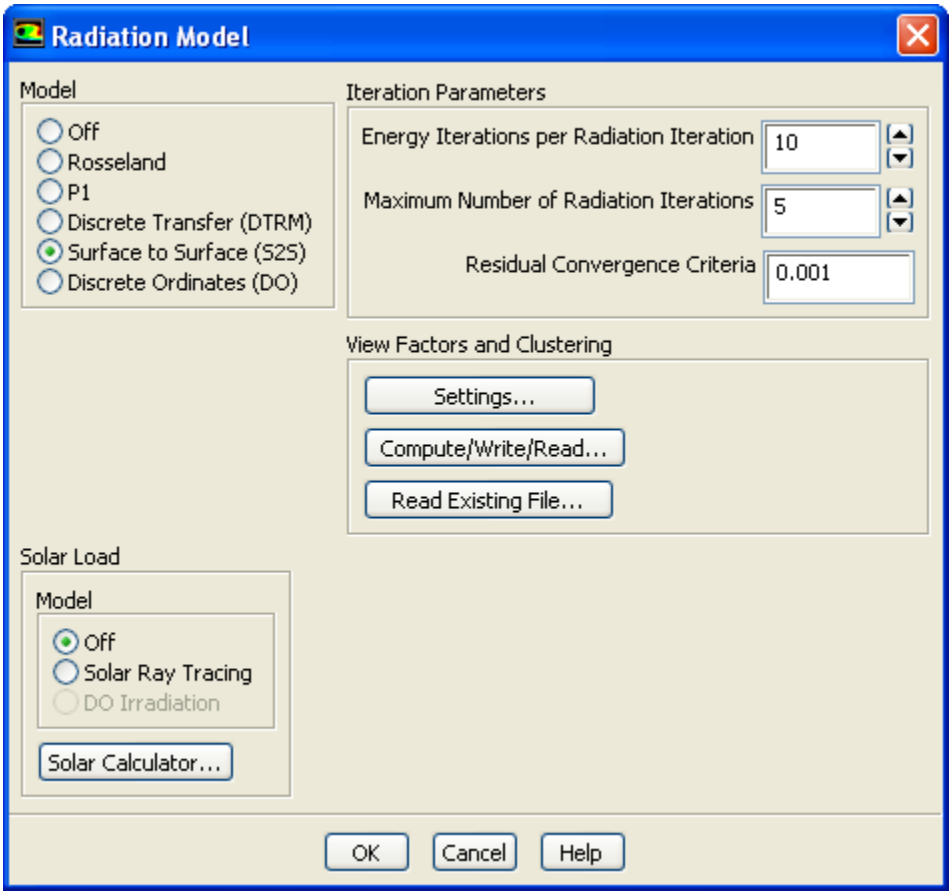
Once a ray file has been created or read in manually, you can click on the **Display Clusters** button in the **DTRM Rays** dialog box to graphically display the clusters in the domain. See *Displaying Rays and Clusters for the DTRM* (p. 738) for additional information about displaying rays and clusters.

14.3.4. Setting Up the S2S Model

When you select the **Surface to Surface (S2S)** model, the **Radiation Model** dialog box will expand to show additional parameters (see *Figure 14.10* (p. 710)). In these additional group boxes, you will set the solution parameters (see *S2S Solution Parameters* (p. 734) for further details), access the view factors and clustering settings, and compute the view factors for your problem or read previously computed view factors into ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.10 The Radiation Model Dialog Box (S2S Model)



The S2S radiation model is very expensive in terms of computation effort and memory requirements when there are a large number of radiating surfaces. You can reduce the number of radiating surfaces by grouping faces together to form surface clusters. The surface cluster information (coordinates and connectivity of the nodes, surface cluster IDs) is used by ANSYS FLUENT in the radiosity calculations and to compute the view factors.

Important

- You should recreate the surface cluster information whenever you do anything that changes the mesh, such as:
  - change the type of a boundary zone
  - reorder the mesh
  - scale the mesh

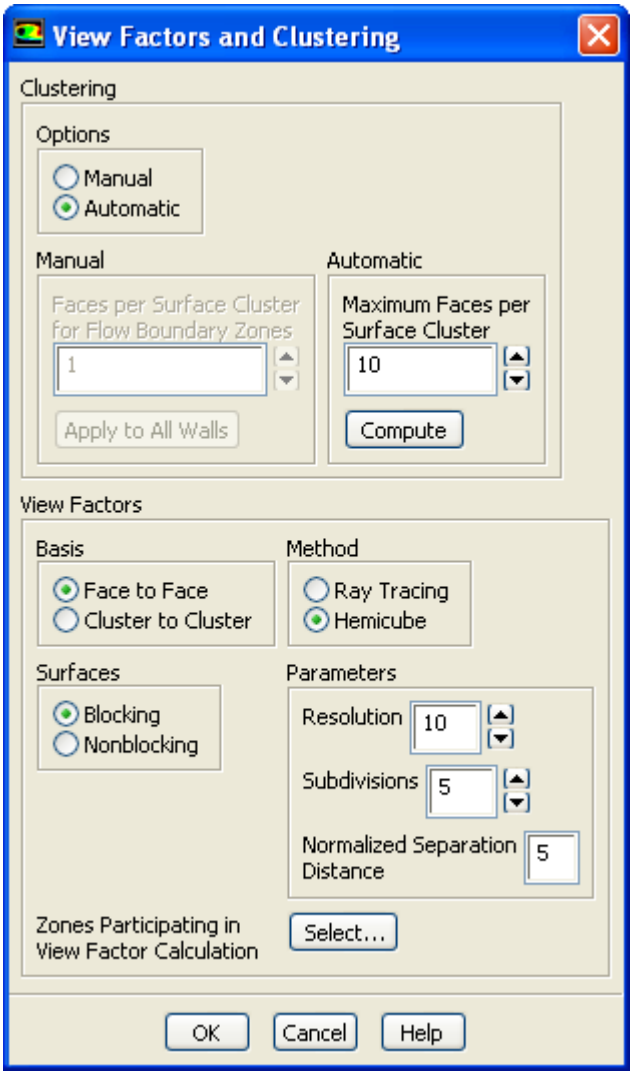
  Note that you do *not* need to recalculate view factors after shell conduction at any wall has been enabled or disabled. See *Thermal Boundary Conditions at Walls* (p. 309) for more information about shell conduction.
- ANSYS FLUENT will warn you to recreate the cluster/view factor file if a boundary zone has been changed from a wall to an internal wall (or vice versa), or if a boundary zone has been merged, separated, or fused.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.4.1. View Factors and Clustering Settings

You can use the **View Factors and Clustering** dialog box (*Figure 14.11* (p. 711)) to define how the surface clusters are formed and how the view factors are calculated for the S2S model. To open this dialog box, click **Settings...** in the **View Factors and Clustering** group box in the *Radiation Model Dialog Box* (p. 1657) (*Figure 14.10* (p. 710)) or use the **File/Write/Surface Clusters...** menu item.

Figure 14.11 The View Factors and Clustering Dialog Box



14.3.4.1.1. Forming Surface Clusters

You can set the number of faces per surface cluster (FPSC) for each flow boundary (i.e., exhaust fan, inlet vent, intake fan, outlet vent, mass-flow inlet, pressure far-field, pressure inlet, pressure outlet, outflow, and velocity inlet boundary) and wall that is adjacent to a fluid zone; in this way, you can control the number of radiating surfaces and (if you select **Cluster to Cluster** for **Basis**) view factor surfaces. By default, the FPSC value is set to 1 everywhere, so the number of surface clusters (radiating/view factor surfaces) will be equal to the number of boundary faces. For small 2D problems, this is an acceptable number. For larger problems, you may want to reduce the number of surface clusters to reduce both the size of the view factor file and the memory requirement. Such a reduction in the number of clusters, however, comes at the cost of some accuracy. See Clustering in the Theory Guide for details about clustering.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

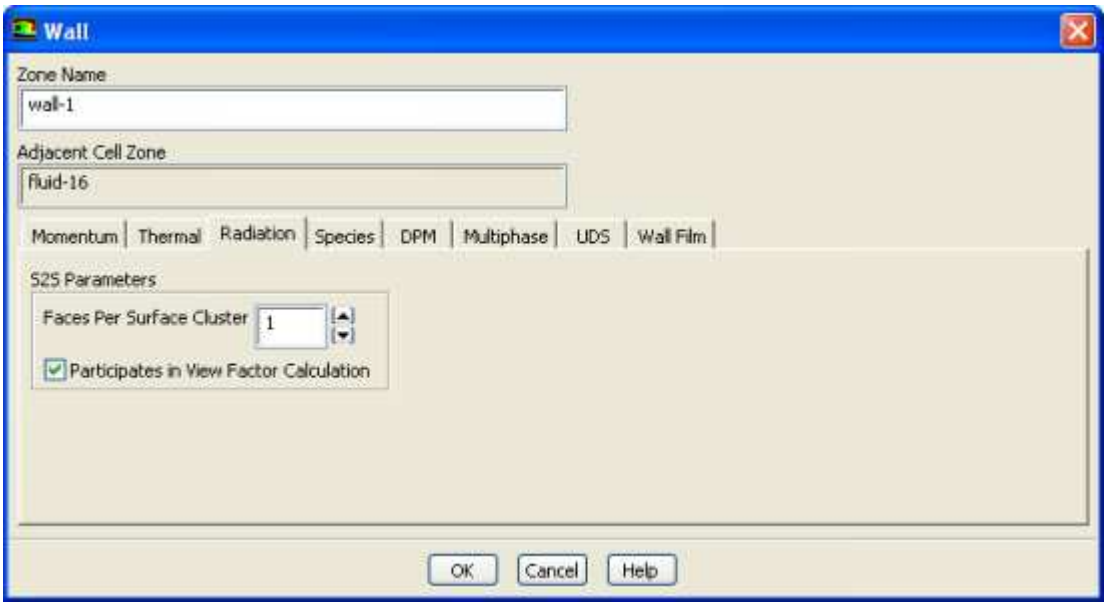
If you plan to use the cluster to cluster basis for the view factor calculation, be sure that the FPSC values are appropriate for both the radiosity calculation and the view factors.

The **Clustering** group box in the **View Factors and Clustering** dialog box (*Figure 14.11* (p. 711)) provides two methods for revising the default FPSC values:

- manual
- automatic

If you select **Manual** in the **Options** group box, you can specify an FPSC value for all flow boundaries in the **Faces per Surface Cluster for Flow Boundary Zones** number-entry box in the **Manual** group box. If you then click **Apply to All Walls**, you will apply this value to all wall zones that are adjacent to fluid zones as well. For a wall that requires a different FPSC value (i.e., you may want a lower value for walls in critical areas, and higher values in non-critical areas), you will need to open the boundary condition dialog box (*Figure 14.12* (p. 712)) for that particular wall and modify the **Faces Per Surface Cluster** in the **S2S Parameters** group box of the **Radiation** tab. Note that the **Radiation** tab also allows you to specify whether the wall participates in the view factor calculation, as described in *Specifying Boundary Zone Participation* (p. 715).

Figure 14.12 The Wall Dialog Box



Important

The **Faces Per Surface Cluster** number-entry box will not be visible in the GUI on wall boundary zones that are attached to a solid.

Using the manual method to specify the FPSC values for walls can become very cumbersome if the model involves a large number of radiating faces, which is typically the case in underhood models. In such circumstances, it is recommended that you use the automatic clustering method instead. In this method, different FPSC values are assigned to the walls automatically, based on the distance of the walls from and the FPSC values of the walls that are defined as critical. The steps you will need to take are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Automatic** from the **Options** list in the **View Factors and Clustering** dialog box.
2. For each wall that you deem critical, perform the following actions in the **Wall** dialog box (*Figure 14.12* (p. 712)):
   a. Click the **Radiation** tab.
   b. Enter an appropriate value for **Faces Per Surface Cluster** in the **S2S Parameters** group box.
   c. Enable the **Critical Zone** option.
   d. Click **OK** to close the **Wall** dialog box.
3. Enter the **Maximum Faces per Surface Cluster** value in the **View Factors and Clustering** dialog box and click the **Compute** button. ANSYS FLUENT will automatically calculate and update the **Faces Per Surface Cluster** values for all **Wall** dialog boxes adjacent to fluid zones that do not have **Critical Zone** enabled, without computing the clusters.
4. You can check the automatically assigned FPSC values by opening the boundary condition dialog box of any non-critical wall of interest and examining the value for **Faces Per Surface Cluster** in the **S2S Parameters** group box of **Radiation** tab. You can manually modify the value for **Faces Per Surface Cluster** as necessary.

14.3.4.1.1.1. Setting the Split Angle for Clusters

Whether you set the FPSC value manually or automatically, you have the option of modifying the cutoff or "split" angle between adjacent face normals for the purpose of controlling surface clustering. The split angle sets the limit for which adjacent faces are clustered. A smaller split angle allows for a better representation of the view factor. By default, no surface cluster will contain any face that has a face normal greater than 20°. To modify the value of this parameter, you can use the `split-angle` text command:

`define` → `models` → `radiation` → `s2s-parameters` → `split-angle`

or

`file` → `write-surface-clusters` → `split-angle`

14.3.4.1.2. Setting Up the View Factor Calculation

You can control many aspects of how the view factors are calculated for your S2S model: how surfaces are defined; the computational method and related parameters; whether surface blocking will be accounted for; and which boundary zones will participate in the calculation. All of these controls are available in the **View Factors** group box in the **View Factors and Clustering** dialog box (*Figure 14.11* (p. 711)), and are described in the sections that follow.

14.3.4.1.2.1. Selecting the Basis for Computing View Factors

ANSYS FLUENT allows two ways to define the surfaces used for the view factor calculation, as described in Clustering and View Factors in the Theory Guide. If you want the surfaces to be the boundary faces of the mesh, select **Face to Face** for **Basis** in the **View Factor** group box of the **View Factors and Clustering** dialog box. This is the default selection.

Alternatively, you can select **Cluster to Cluster** for **Basis**, in order to reduce the computational expense and storage requirements. In this case, the surfaces used to calculate the view factors are the clusters defined by the settings in the **Cluster** group box of the **View Factors and Clustering** dialog box. The reduction in computational time will be proportional to the number of surface clusters used in the view factor calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The trade-offs of using the cluster to cluster basis for the calculation are that the accuracy may decrease, and the following limitations apply:

- The mesh must be 3D.
- Only the hemicube method can be used, as described in *Selecting the Method for Computing View Factors* (p. 714).
- You cannot subdivide the faces as part of the hemicube method parameters, which can cause the view factors to be overestimated.
- Polyhedral meshes are restricted to 1 face per surface cluster.

14.3.4.1.2.2. Selecting the Method for Computing View Factors

ANSYS FLUENT provides two methods for computing view factors: the ray tracing method (which is selected by default) and the hemicube method. The following limitations apply:

- The hemicube method is available only for 3D and axisymmetric cases.
- The ray tracing method should not be used when any of the zones are defined as periodic, as this type of zone is not currently supported.
- The ray tracing method is only available when **Face to Face** is selected for **Basis**.

The hemicube method uses a differential area-to-area method and calculates the view factors on a row-by-row basis. The view factors calculated from the differential areas are summed to provide the view factor for the whole surface. This method originated from the use of the radiosity approach in the field of computer graphics [*16*] (p. 2209).

To use the hemicube method to compute the view factors, select **Hemicube** from the **Method** list in the **View Factors and Clustering** dialog box. It is recommended that you use the hemicube method for large, complex models with few obstructing surfaces between the radiating surfaces.

The hemicube method is based upon three assumptions about the geometry of the surfaces: aliasing, visibility, and proximity. To validate these assumptions, you can specify three different hemicube parameters, which can help you obtain better accuracy in calculating view factors. In most cases, however, the default settings will be sufficient.

- Aliasing—The true projection of each visible face onto the hemicube can be accurately accounted for by using a finite-resolution hemicube. As described previously, the faces are projected onto a hemicube. Because of the finite resolution of the hemicube, the projected areas and resulting view factors may be overestimated or underestimated. Aliasing effects can be reduced by increasing the value of the **Resolution** of the hemicube in the **Parameters** group box.
- Visibility—The visibility between any two faces does not change. In some cases, face i has a complete view of face k from its centroid, but some other face j occludes much of face k from face i. In such a case, the hemicube method will overestimate the view factor between face i and face k calculated from the centroid of face i. This error can be reduced by subdividing face i into smaller subfaces. You can specify the number of subfaces by entering a value for **Subdivisions** in the **Parameters** group box. Note that you cannot subdivide the faces when **Cluster to Cluster** is selected for **Basis**.
- Proximity—The distance between faces is great compared to the effective diameter of the faces. The proximity assumption is violated whenever faces are close together in comparison to their effective diameter or are adjacent to one another. In such cases, the distances between the centroid of one face and all points on the other face vary greatly. Since the view factor dependence on distance is non-linear, the result is a poor estimate of the view factor.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Parameters** group box, you can set a limit for the **Normalized Separation Distance**, which is the ratio of the minimum face separation to the effective diameter of the face. If the computed normalized separation distance is less than the specified value, the face will then be divided into a number of subfaces until the normalized distances of the subfaces are greater than the specified value. Alternatively, you can specify the number of subfaces to create for such faces by entering a value for **Subdivisions**.

While the hemicube method projects radiating surfaces onto a hemicube, the ray tracing method instead traces rays through the centers of every hemicube face to determine which surfaces are visible through that face. Also, the ray tracing method is OpenMP parallelized and will therefore use all available processors when performing the ray tracing calculations (for further details, visit `http://www.openmp.org`). As a result, the calculation time is reduced for large, complex geometries that have obstructions between the radiating surfaces (such as automotive underhood simulations). Note that the ray tracing method does not subdivide the faces (as can be done when using the hemicube method by setting the **Subdivisions** or **Normalized Separation Distance** parameters), and so the view factors may be less accurate than those calculated using the hemicube method for surfaces that have a normalized separation distance less than 5.

To use the ray tracing method to compute the view factors, select **Ray Tracing** from the **Method** list in the **View Factors and Clustering** dialog box. You can adjust the value of the **Resolution** in the **Parameters** group box in order to reduce the impact of aliasing effects, as described previously.

14.3.4.1.2.3. Accounting for Blocking Surfaces

View factor calculations depend on the geometric orientations of surface pairs with respect to each other. Two situations may be encountered when examining surface pairs:

- If there is no obstruction between the surface pairs under consideration, then they are referred to as “nonblocking” surfaces.
- If there is another surface blocking the views between the surfaces under consideration, then they are referred to as “blocking” surfaces. Blocking will change the view factors between the surface pairs and require additional checks to compute the correct value of the view factors.

For cases with blocking surfaces, select **Blocking** from the **Surfaces** list in the **View Factors and Clustering** dialog box. For cases with nonblocking surfaces, you can choose either **Blocking** or **Nonblocking** without affecting the accuracy. However, it is better to choose **Nonblocking** for such cases, as it takes less time to compute.

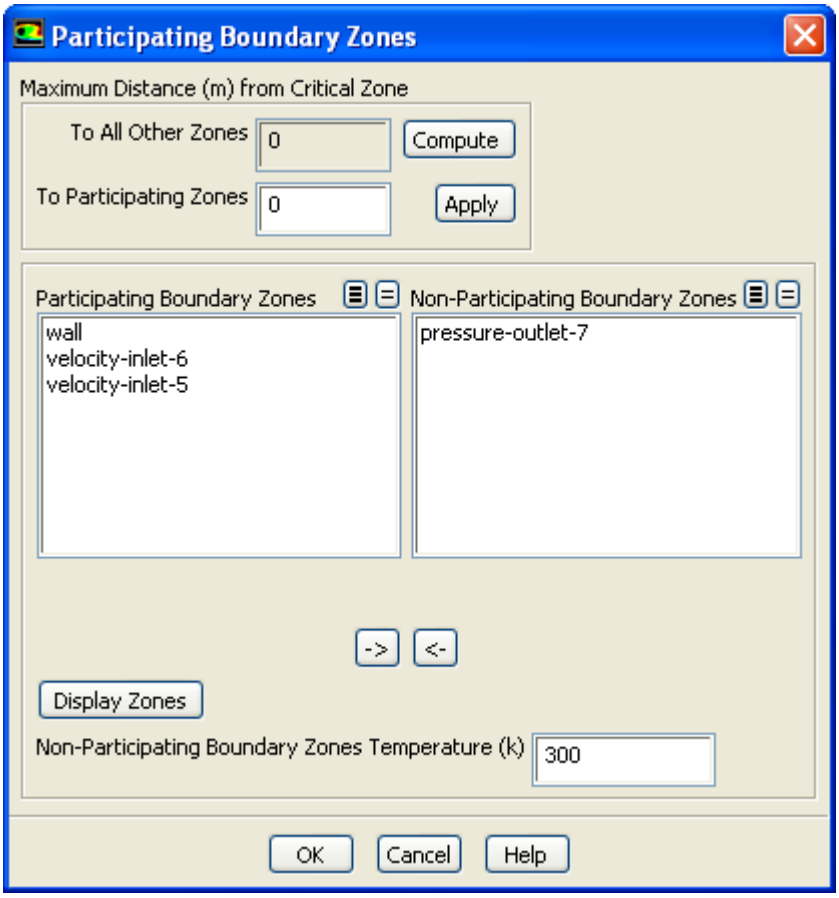
14.3.4.1.2.4. Specifying Boundary Zone Participation

You can choose to exclude walls and inlet and exit boundaries from participating in the view factor calculation. If you are unsure whether it is necessary to calculate view factors for a particular boundary zone ahead of time, it is recommended that you allow it to participate; you can always reverse this decision as part of a future calculation run.

There are two ways in which you can enable/disable the participation of walls and inlet and exit boundaries in the view factor calculation. One of those ways is to use the **Participates in View Factor Calculation** option in the **Radiation** tab of the boundary condition dialog box. The other method is to use the **Participating Boundary Zones** dialog box (*Figure 14.13* (p. 716)), which is accessed by clicking the **Select...** button next to the **Zones Participating in View Factor Calculation** label in the **View Factors and Clustering** dialog box. For cases that are comprised of a very large number of zones, such as underhood applications, the latter method is recommended.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.13 The Participating Boundary Zones Dialog Box



The **Participating Boundary Zones** dialog box allows you to easily specify those zones that are participating or non-participating without having to visit the boundary conditions dialog box of each zone. For cases that have a small number of boundary zones, you can simply select the zones that you do not want to participate in the view factor calculation from the **Participating Boundary Zones** list and click the arrow button that points to the left, so that the zones are moved to the **Non-Participating Boundary Zones** list; if you make an error, you can always reverse this process (i.e., select a zone in the **Non-Participating Boundary Zones** list and click the arrow button that points to the right). This can be cumbersome for cases that have a large number of boundary zones, and so the following procedure is recommended instead:

1. Make sure that your clustering options (see *Forming Surface Clusters* (p. 711)) are appropriate for your view factor settings:

   a. Select **Automatic** from the **Options** list in the **Clustering** group box of the **View Factors and Clustering** dialog box, as this enables some GUI items in the **Participating Boundary Zones** dialog box.

Important

Note that if you do not want to use the **Automatic** option for the clustering, you can revert to the **Manual** option after you are done using the **Participating Boundary Zones** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Verify that the walls you specified as critical (by enabling the **Critical Zone** option in the **Radiation** tab of the boundary condition dialog box) also correspond to those that are critical for the view factor calculation. You must have specified at least one critical zone for the steps that follow.

2. Click the **Compute** button in the **Maximum Distance from Critical Zone** dialog box, to update the value displayed for **To All Other Zones**. This value represents the maximum distance between the centroids of a critical zone and a non-critical zone in the mesh; it is for information purposes only, and cannot be edited in this dialog box.
3. Based on the value displayed for **To All Other Zones**, enter a threshold value for **To Participating Zones**. The value you enter will specify the maximum distance allowed between the centroids of a critical zone and a zone that participates in the view factor calculation. Then click the **Apply** button to move all zones beyond this distance into the **Non-Participating Boundary Zones** list.
4. Review the zones displayed in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists. If necessary, select zones in these lists and use the arrow buttons to move them to the appropriate list, as described previously.

If at any point you want to visually identify zones displayed in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists, select the zones and click the **Display Zones** button. Only the selected zones will be displayed in the graphics window.

If any zones are displayed in the **Non-Participating Boundary Zones** list, ensure to enter an appropriate temperature for **Non-Participating Boundary Zones Temperature**. In most cases the appropriate value is the ambient temperature, which by default is assumed to be 300 K.

After you have specified which zones do not participate in view factor calculation and set their temperature, click **OK** to store the settings and close the **Participating Boundary Zones** dialog box. You can then proceed to computing the view factors, as described in the section that follows.

14.3.4.2. Computing View Factors

ANSYS FLUENT can compute the view factors for your problem in the current session and save them to a file for use in the current session and future sessions. Alternatively, you can save the surface cluster information and view factor parameters to a file, calculate the view factors outside ANSYS FLUENT, and then read the view factors into ANSYS FLUENT. These methods for computing view factors are described in the following sections.

Important

For large meshes or complex models, it is recommended that you calculate the view factors outside ANSYS FLUENT and then read them into ANSYS FLUENT before starting your simulation.

14.3.4.2.1. Computing View Factors Inside ANSYS FLUENT

To compute view factors in your current ANSYS FLUENT session, you must first set the parameters for the view factor calculation in the **View Factors and Clustering** dialog box (see *View Factors and Clustering Settings* (p. 711) for details) and click **OK** to save them. When you have set the view factor and surface cluster parameters, click the **Compute/Write/Read...** button in the **View Factors and Clustering** group box of the **Radiation Model** dialog box. A **Select File** dialog box will open, prompting you for the name of the file in which ANSYS FLUENT should save the surface cluster information and the view factors. After you have specified the file name, ANSYS FLUENT will write the surface cluster information to the file. ANSYS FLUENT will use the surface cluster information to compute the view factors, save the view factors to the same file, and then

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatically read the view factors. The ANSYS FLUENT console will report the status of the view factor calculation. For example:

```
Completed 25% calculation of viewfactors

Completed 50% calculation of viewfactors

Completed 75% calculation of viewfactors

Completed 100 % calculation of viewfactors
```

Important

- The view factor file format for this version of ANSYS FLUENT is known as the compressed row format (CRF) which is a more efficient way of writing view factors than in prior versions of ANSYS FLUENT. In the CRF format, only non-zero view factors with their associated cluster IDs are stored to the file. This reduces the size of the `.s2s` file, and reduces the time it takes to read the file into ANSYS FLUENT. While the CRF file format is the default, you can still use the older file format if necessary. Contact your support engineer for more information.
- View factors using ray tracing is computed with the MPI/OpenMP hybrid model for parallel runs. By default, all the available cores within a machine are used as OpenMP threads. You can overwrite this option by setting the desired number of cores or processors to be used as OpenMP threads using the `(%set-openmp-threads N)` scheme command, where N is the number of cores or processors to be used as OpenMP threads. You can query the OpenMP threads being set using the `(%get-openmp-threads)` scheme command.

Note

For cases which use the S2S model and contain non-conformal interfaces, intersection of the interface zones will occur during the initialization process, which will cause relabeling of faces throughout the domain. If the view factors are computed before initialization, a mismatch with the case may occur after initialization. Therefore, for such cases, perform the following steps:

- Read the case.
- Disable the S2S model if it is already enabled.
- Initialize the case.
- Save the case and data and read it back. This step is needed since you could unintentionally read the case before initialization and try to read the view factor file, which was generated after the intersections were taken.
- Enable the S2S model.
- Compute the view factors and proceed as usual.

14.3.4.2.2. Computing View Factors Outside ANSYS FLUENT

To compute view factors outside ANSYS FLUENT, you must save the surface cluster information and view factor parameters to a file.

File → **Write** → **Surface Clusters...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT will open the **View Factors and Clustering** dialog box, where you will set the view factor and surface cluster parameters (see *View Factors and Clustering Settings* (p. 711) for details). When you click **OK** in the **View Factors and Clustering** dialog box, a **Select File** dialog box will open, prompting you for the name of the file in which ANSYS FLUENT should save the surface cluster information and view factor parameters to the file. If the specified **Filename** ends in `.gz` or `.Z`, appropriate file compression will be performed.

To calculate the view factors outside ANSYS FLUENT, enter one of the following commands:

- For the serial solver:

  `utility viewfac` *inputfile*

  where *inputfile* is the filename, or the correct path to the filename, for the surface cluster information and view factor parameters file that you saved from ANSYS FLUENT. You can then read the view factors into ANSYS FLUENT, as described below.

- For the parallel solver:
  - Viewfac utility module:

  `utility viewfac -tnprocs [-pinterconnect] [-mpi=mpi type] -cnf=` *hosts_file inputfile*

  where *inputfile* is the filename for which you will need to provide the full path.

  - Raytracing utility module:

  `utility raytracing -tnprocs [-pinterconnect] [-mpi=mpi type] -cnf=` *hosts_file* `[-ompthreads=N]` *inputfile*

  where *inputfile* is the filename for which you will need to provide the full path.

  On Windows:
  `-tnprocs` specifies the number of processes to use. When the `-cnf` option is present, the hosts file argument is used to determine which computers to use for the parallel job. For example, if there are 8 computers listed in the hosts file and you want to run a job with 4 processes, set `nprocs` to 4 (i.e. `-t4`) and ANSYS FLUENT will use the first 4 machines listed in the hosts file. Note that this does not apply to the Compute Cluster Server (CCS).
  `-pinterconnect` (optional) specifies the type of interconnect. The ethernet interconnect is used by default if the option is not explicitly specified. See *Table 34.1: Supported Interconnects for the Windows Platform* (p. 1596), *Table 34.2: Available MPIs for Windows Platforms* (p. 1596), and *Table 34.3: Supported MPIs for Windows Architectures (Per Interconnect)* (p. 1597) for more information.
  `-mpi=mpi type` (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect (HP MPI) will be used (the use of the default MPI is recommended). The available MPIs for Windows are shown in *Table 34.2: Available MPIs for Windows Platforms* (p. 1596).
  `-cnf=hosts` file specifies the hosts file, which contains a list of the computers on which you want to run the parallel job. You will need to supply the full pathname to the file, or you can use an explicit host list such as `-cnf=host1,host2,...,hostn`
  `-ompthreads=N` (optional) where N is the desired number of OpenMP threads to be used per machine.

  On Linux/UNIX:
  `-tnprocs` specifies the number of processes to used. When the `-cnf` option is present, the hosts file argument is used to determine which computers to use for the parallel job. For example, if there

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are 10 computers listed in the hosts file and you want to run a job with 5 processes, set `nprocs` to 5 (i.e. `-t5`) and ANSYS FLUENT will use the first 5 machines listed in the hosts file.
`-pinterconnect` (optional) specifies the type of interconnect. The ethernet interconnect is used by default if the option is not explicitly specified. See *Table 34.4: Supported Interconnects for Linux/UNIX Platforms (Per Platform)* (p. 1601), *Table 34.5: Available MPIs for Linux/UNIX Platforms* (p. 1601), and *Table 34.6: Supported MPIs for Linux/UNIX Architectures (Per Interconnect)* (p. 1601) for more information.
`-mpi=mpi` type (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect will be used (the use of the default MPI is recommended). The available MPIs for Linux/UNIX are shown in *Table 34.5: Available MPIs for Linux/UNIX Platforms* (p. 1601).
`-cnf=hosts` file specifies the hosts file, which contains a list of the computers on which you want to run the parallel job, you will need to supply the full pathname to the file.
`-ompthreads=N` (optional) where N is the desired number of OpenMP threads to be used per machine.

14.3.4.3. Reading View Factors into ANSYS FLUENT

If the view factors for your problem have already been computed (either inside or outside ANSYS FLUENT) and saved to a file, you can read them into ANSYS FLUENT. To read in the view factors, click **Read Existing File...** button in the **View Factors and Clustering** group box of the **Radiation Model** dialog box. A **Select File** dialog box will open where you can specify the name of the file containing the view factors. You can also manually specify the view factors file, using the **File/Read/View Factors...** menu item.

Important

While the previous `.s2s` view factor file format can still be read seamlessly into ANSYS FLUENT, there is now a more efficient compressed row format (CRF) that can be read into ANSYS FLUENT (see the section on Computing View Factors Inside ANSYS FLUENT). You can take advantage of the reduced size of the CRF file and thus the reduced time it takes to read the file into ANSYS FLUENT, by converting the existing old file format to the new format (without having to recompute the view factors) using the following command at the command prompt in your working directory:

```
utility viewfac -c1 -o new.s2s.gz old.s2s.gz
```

where `new.s2s.gz` is the CRF format to which you want the old file format (`old.s2s.gz`) converted.

14.3.5. Setting Up the DO Model

14.3.5.1. Angular Discretization

When you select the **Discrete Ordinates** model, the **Radiation Model** dialog box will expand to show inputs for **Angular Discretization** (see *Figure 14.7* (p. 705)). In this section, you will set parameters for the angular discretization and pixelation described in Angular Discretization and Pixelation in the Theory Guide.

Theta Divisions (N_θ) and **Phi Divisions** (N_ϕ) will define the number of control angles used to discretize each octant of the angular space (see Figure 5.3: "Angular Coordinate System" in the Theory Guide). Note that higher levels of discretization are recommended for problems where specular exchange of radiation is important to increase the likelihood of the correct beam direction being captured. For a 2D model, ANSYS FLUENT will solve only 4 octants (due to symmetry); thus, a total of $4N_\theta N_\phi$ directions $\vec{s}$ will be solved. For a 3D model, 8 octants are solved, resulting in $8N_\theta N_\phi$ directions $\vec{s}$. By default, the number of **Theta Divisions** and the number of **Phi Divisions** are both set to 2. For most practical problems, these settings are acceptable,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

however, a setting of 2 is considered to be a coarse estimate. Increasing the discretization of **Theta Divisions** and **Phi Divisions** to a minimum of 3, or up to 5, will achieve more reliable results. A finer angular discretization can be specified to better resolve the influence of small geometric features or strong spatial variations in temperature, but larger numbers of **Theta Divisions** and **Phi Divisions** will add to the cost of the computation.

Theta Pixels and **Phi Pixels** are used to control the pixelation that accounts for any control volume overhang (see Figure 5.7: "Pixelation of Control Angle" in the Theory Guide and the figures and discussion preceding it). For problems involving gray-diffuse radiation, the default pixelation of 1×1 is usually sufficient. For problems involving symmetry, periodic, specular, or semi-transparent boundaries, a pixelation of 3×3 is recommended and will achieve acceptable results. The computational effort, as a result of increasing the pixelation, is less than the computational effort caused by increasing the divisions. You should be aware, however, that increasing the pixelation adds to the cost of computation.

Important

Note that pixelations are applied to boundary faces by default.

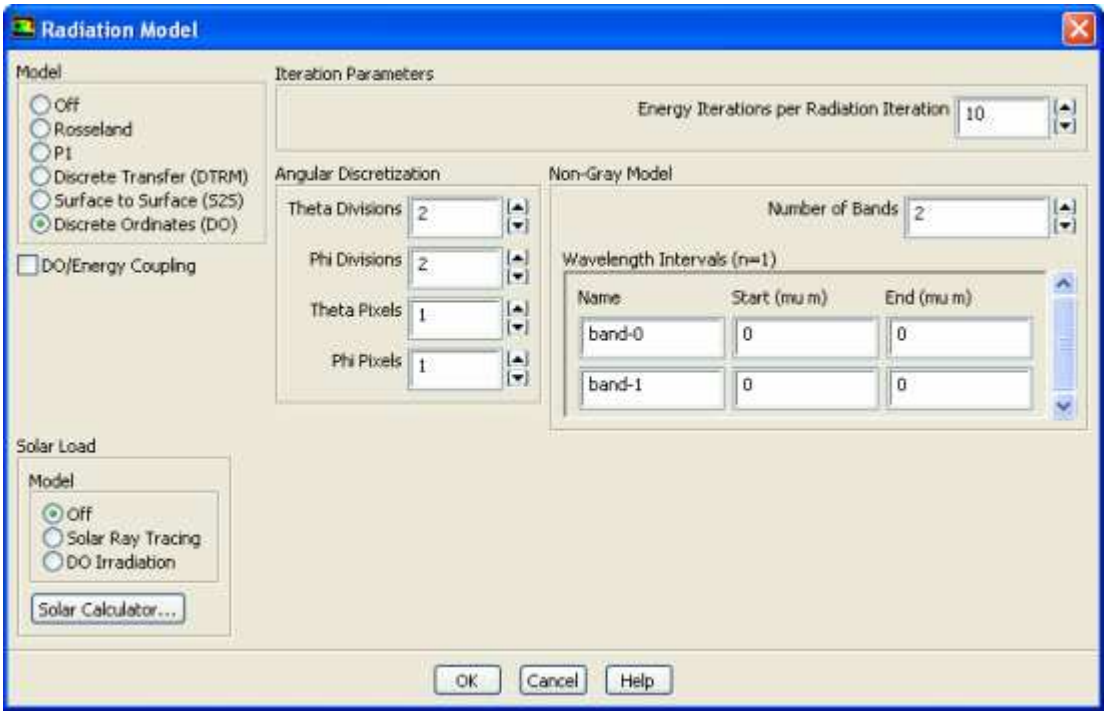
14.3.5.2. Defining Non-Gray Radiation for the DO Model

If you want to model non-gray radiation using the DO model, you can specify the **Number of Bands** (N) under **Non-Gray Model** in the expanded **Radiation Model** dialog box (*Figure 14.14* (p. 722)). For a 2D model, ANSYS FLUENT will solve $4N_\theta N_\phi N$ directions. For a 3D model, $8N_\theta N_\phi N$ directions will be solved. By default, the **Number of Bands** is set to zero, indicating that only gray radiation will be modeled. Because the cost of computation increases directly with the number of bands, you should try to minimize the number of bands used. In many cases, the absorption coefficient or the wall emissivity is effectively constant for the wavelengths of importance in the temperature range of the problem. For such cases, the gray DO model can be used with little loss of accuracy. For other cases, non-gray behavior is important, but relatively few bands are necessary. For typical glasses, for example, two or three bands will frequently suffice.

When a non-zero **Number of Bands** is specified, the **Radiation Model** dialog box will expand once again to show the **Wavelength Intervals** (*Figure 14.14* (p. 722)). You can specify a **Name** for each wavelength band, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum ($n = 1$). ANSYS FLUENT will automatically account for the refractive index in setting band limits for media with n different from unity.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.14 The Radiation Model Dialog Box (Non-Gray DO Model)



The frequency of radiation remains constant as radiation travels across a semi-transparent interface. The wavelength, however, changes such that $n\lambda$ is constant. Thus, when radiation passes from a medium with refractive index n_1 to one with refractive index n_2, the following relationship holds:

$$n_1\lambda_1 = n_2\lambda_2 \qquad \textbf{(14–7)}$$

Here λ_1 and λ_2 are the wavelengths associated with the two media. It is conventional to specify the wavelength rather than frequency. ANSYS FLUENT requires you to specify wavelength bands for an equivalent medium with $n = 1$.

For example, consider a typical glass with a step jump in the absorption coefficient at a cut-off wavelength of λ_c. The absorption coefficient is a_1 for $\lambda \leq \lambda_c \mu$m and a_2 for $\lambda \lambda_c \mu$m. The refractive index of the glass is n_g. Since $n\lambda$ is constant across a semi-transparent interface, the equivalent cut-off wavelength for a medium with $n = 1$ is $n_g \lambda_c$ using *Equation 14–7* (p. 722). You should choose two bands in this case, with the limits 0 to $n_g \lambda_c$ and $n_g \lambda_c$ to 100. Here, the upper wavelength limit has been chosen to be a large number, 100, in order to ensure that the entire spectrum is covered by the bands. When multiple materials exist, you should convert all the cut-off wavelengths to equivalent cut-off wavelengths for an $n = 1$ medium, and choose the band boundaries accordingly.

The bands can have different widths and need not be contiguous. You can ensure that the entire spectrum is covered by your bands by choosing $\lambda_{min} = 0$ and $n\lambda_{max} T_{min} \geq 50{,}000$. Here λ_{min} and λ_{max} are the minimum and maximum wavelength bounds of your wavelength bands, and T_{min} is the minimum expected temperature in the domain.

ANSYS FLUENT allows you to use a user-defined function (UDF) to modify the emissivity weighting factor $F(0 \rightarrow n\lambda_2 T) - F(0 \rightarrow n\lambda_1 T)$ (which otherwise defaults to the black body emission factor obtained from

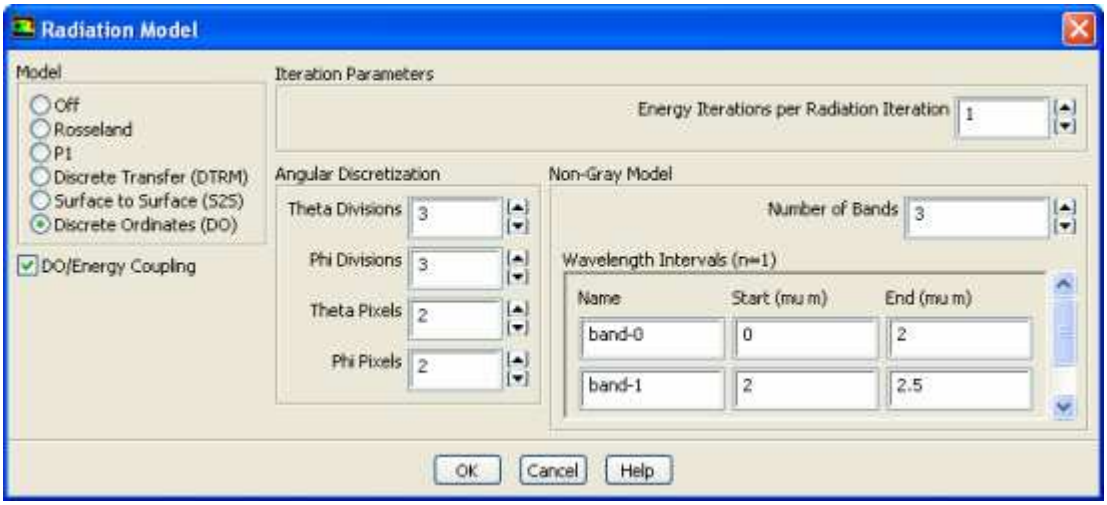
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a standard Planck distribution). The emissivity weighting factor appears in the emission term of the radiative transfer equation for the non-gray model, as shown in Equation 5–61 in the Theory Guide. For more information, see DEFINE_EMISSIVITY_WEIGHTING_FACTOR in the UDF Manual.

14.3.5.3. Enabling DO/Energy Coupling

For applications involving optical thicknesses greater than 10, you can enable the **DO/Energy Coupling** option in the **Radiation Model** (*Figure 14.15* (p. 723)) in order to couple the energy and intensity equations at each cell, solving them simultaneously. This approach accelerates the convergence of the finite volume scheme for radiative heat transfer and can be used with the gray or non-gray radiation model.

Figure 14.15 The Radiation Model Dialog Box with DO/Energy Coupling Enabled



Important

This option should not be used when the shell conduction model is enabled.

14.3.6. Defining Material Properties for Radiation

When you are using the P-1, DO, or Rosseland radiation model in ANSYS FLUENT, you should be sure to define both the absorption and scattering coefficients of the fluid in the **Create/Edit Materials** dialog box. Note that you can either enter a constant value for these parameters, or you can specify them using a user-defined function (UDF). For more information, see DEFINE_PROPERTY UDFs in the UDF Manual.

Materials

If you are modeling semi-transparent media using the DO model, you should also define the refractive index for the semi-transparent fluid or solid material. When using the Rosseland model, you can specify the refractive index only for the fluid material. When using the P-1 model, you should define the refractive index for the fluid material only. For the DTRM, you need to define only the absorption coefficient.

If your model includes gas phase species such as combustion products, absorption and/or scattering in the gas may be significant. The scattering coefficient should be increased from the default of zero if the fluid contains dispersed particles or droplets which contribute to scattering. Alternatively, you can specify the scattering coefficient as a user-defined function (UDF). For more information, see DEFINE_PROPERTY UDFs in the UDF Manual.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT allows you to input a composition-dependent absorption coefficient for CO_2 and H_2O mixtures, using the WSGGM. The method for computing a variable absorption coefficient is described in Radiation in Combusting Flows in the Theory Guide. *Radiation Properties* (p. 435) provides a detailed description of the procedures used for input of radiation properties.

14.3.6.1. Absorption Coefficient for a Non-Gray Model

If you are using the non-gray P-1 model or non-gray DO model, you can specify a different constant absorption coefficient for each of the bands used by the gray-band model, as described in *Radiation Properties* (p. 435). You cannot, however, compute a composition-dependent absorption coefficient in each band. If you use the WSGGM to compute a variable absorption coefficient, the value will be the same for all bands. Alternatively, you can specify a user-defined function (UDF) for the absorption coefficient. For more information, see DEFINE_PROPERTY UDFs in the UDF Manual.

14.3.6.2. Refractive Index for a Non-Gray Model

If you are using the non-gray P-1 model or non-gray DO model, you can specify a different constant refractive index for each of the bands used by the gray-band model, as described in *Radiation Properties* (p. 435). You cannot, however, compute a composition-dependent refractive index in each band.

14.3.7. Defining Boundary Conditions for Radiation

When you set up a problem that includes radiation, you will set additional boundary conditions at inlets, exits, and walls. These inputs are described below.

Boundary Conditions

14.3.7.1. Inlet and Exit Boundary Conditions

14.3.7.1.1. Emissivity

When radiation is active, you can define the emissivity at each inlet and exit boundary when you are defining boundary conditions in the associated inlet or exit boundary dialog box (**Pressure Inlet** dialog box, **Velocity Inlet** dialog box, **Pressure Outlet** dialog box, etc.). Enter the appropriate value for **Internal Emissivity**. The default value for all boundary types is 1. Alternatively, you can specify a user-defined function for emissivity. For more information, see DEFINE_PROPERTY UDFs in the UDF Manual.

For the non-gray P-1 model and the non-gray DO model, the specified constant emissivity will be used for all wavelength bands.

Important

The **Internal Emissivity** boundary condition is not available with the Rosseland model.

14.3.7.1.2. Black Body Temperature

ANSYS FLUENT includes an option that allows you to take into account the influence of the temperature of the gas and the walls beyond the inlet/exit boundaries, and specify different temperatures for radiation and convection at inlets and exits. This is useful when the temperature outside the inlet or exit differs considerably from the temperature in the enclosure. For example, if the temperature of the walls beyond the inlet is 2000 K and the temperature at the inlet is 1000 K, you can specify the outside wall temperature to be used

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for computing radiative heat flux, while the actual temperature at the inlet is used for calculating convective heat transfer. To do this, you would specify a radiation temperature of 2000 K as the black body temperature.

Although this option allows you to account for both cooler and hotter outside walls, you must use caution in the case of cooler walls, since the radiation from the immediate vicinity of the hotter inlet or outlet almost always dominates over the radiation from cooler outside walls. If, for example, the temperature of the outside walls is 250 K and the inlet temperature is 1500 K, it might be misleading to use 250 K for the radiation boundary temperature. This temperature might be expected to be somewhere between 250 K and 1500 K; in most cases it will be close to 1500 K. Its value depends on the geometry of the outside walls and the optical thickness of the gas in the vicinity of the inlet.

In the flow inlet or exit dialog box (**Pressure Inlet** dialog box, **Velocity Inlet** dialog box, etc.), select **Specified External Temperature** in the **External Black Body Temperature Method** drop-down list, and then enter the value of the radiation boundary temperature as the **Black Body Temperature**.

Important

- If you want to use the same temperature for radiation and convection, retain the default selection of **Boundary Temperature** as the **External Black Body Temperature Method**.
- The **Black Body Temperature** boundary condition is not available with the Rosseland model.

14.3.7.2. Wall Boundary Conditions for the DTRM, and the P-1, S2S, and Rosseland Models

The DTRM and the S2S, Rosseland, and gray P-1 (i.e., **Number of Bands** is set to zero) models assume all walls to be gray and diffuse. The only radiation boundary condition required in the **Wall** dialog box is the emissivity. For the Rosseland model, the internal emissivity is 1. For the DTRM and the S2S and gray P-1 models, you can enter the appropriate value for **Internal Emissivity** in the **Thermal** tab of the **Wall** dialog box. The default value is 1. Alternatively, you can specify a user-defined function for emissivity. For more information, see DEFINE_PROPERTY UDFs in the UDF Manual.

For the non-gray P–1 model, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box (the default value in each band is 1). Alternatively, you can specify the internal emissivity using a boundary condition parameter (see *Creating a New Parameter* (p. 211)). See *Defining Boundary Conditions for Radiation* (p. 724) for details.

14.3.7.2.1. Boundary Conditions for the S2S Model

When the S2S model is used, you can specify that some of the walls and inlet and exit boundaries are not participating in the view factor calculation. This capability allows you to save time computing the view factors and also reduce the memory required to store the view factor file during the ANSYS FLUENT calculation. See *Specifying Boundary Zone Participation* (p. 715) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- Whenever you revise which the boundary zones participate in the calculation, you will need to recompute the view factors.
- The **Flux Reports** dialog box will not show the exact balance of the **Radiation Heat Transfer Rate** because the radiative heat transfer to those boundaries that do not participate in the view factor calculation is not included.

14.3.7.3. Wall Boundary Conditions for the DO Model

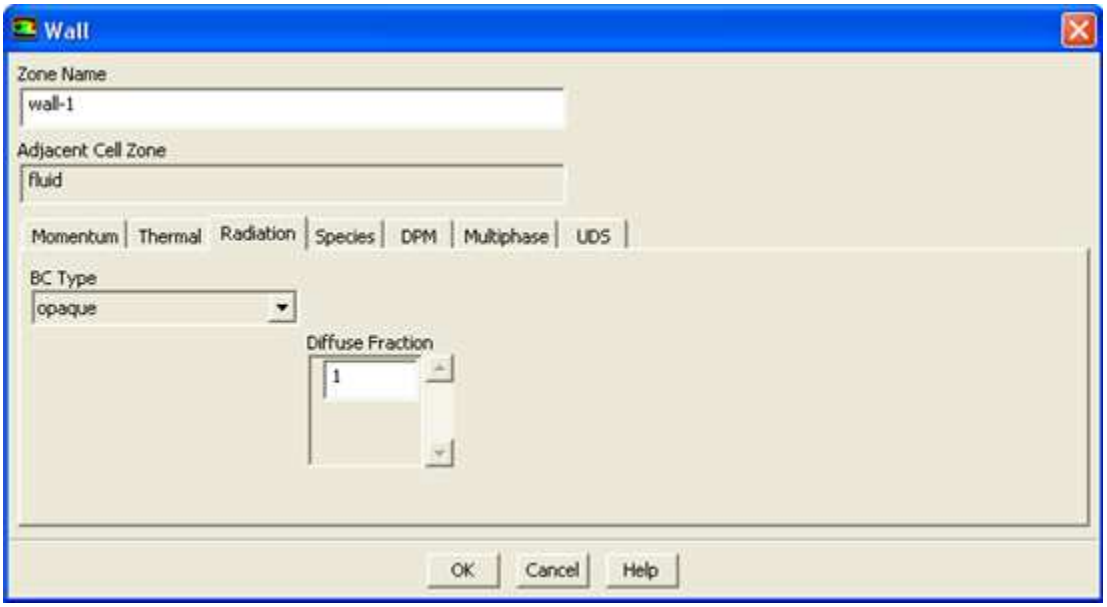
When the DO model is used, you can model opaque walls, as discussed in Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide, as well as semi-transparent walls (Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide).

You can use a diffuse wall to model wall boundaries in many industrial applications since, for the most part, surface roughness makes the reflection of incident radiation diffuse. For highly polished surfaces, such as reflectors or mirrors, the specular boundary condition is appropriate. The semi-transparent boundary condition can be appropriate, for example, when modeling for glass panes in air.

14.3.7.3.1. Opaque Walls

In the **Radiation** tab of the **Wall** dialog box (*Figure 14.16* (p. 726)), select **opaque** in the **BC Type** drop-down list to specify an opaque wall. Opaque walls are treated as gray if gray radiation is being computed, or non-gray if the non-gray DO model is being used. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band.

Figure 14.16 The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall

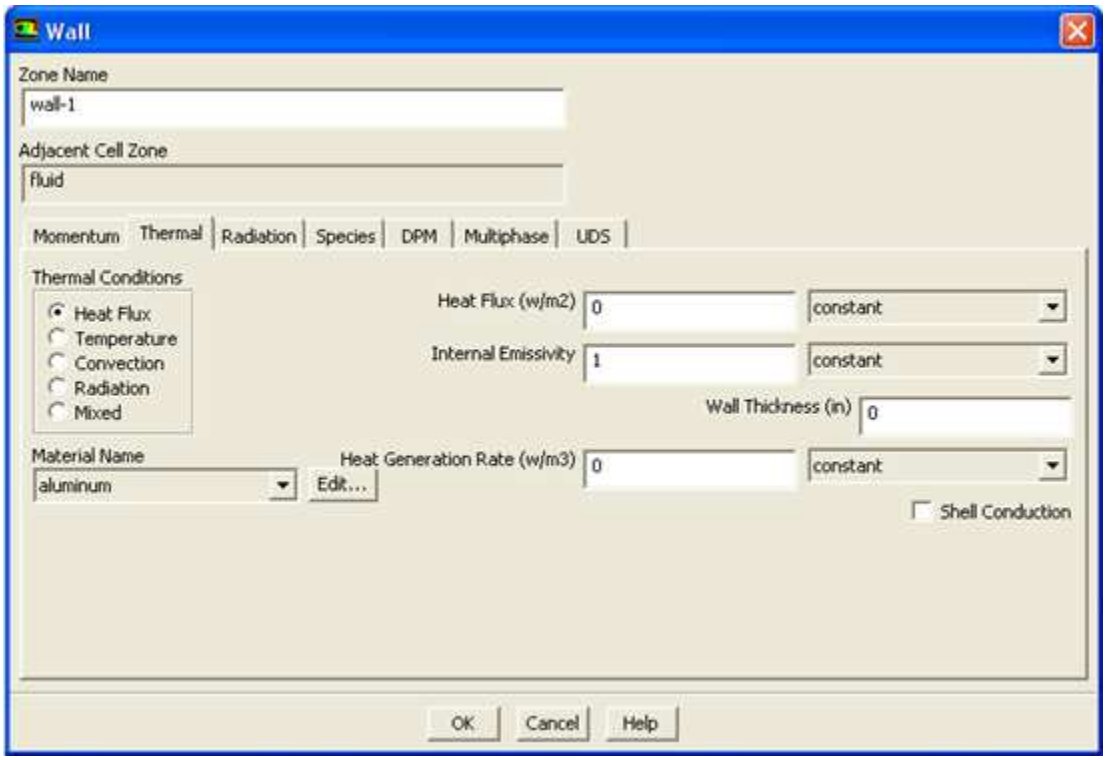


After you have selected **opaque** as the **BC Type**, you can specify the fraction of reflected radiation flux that is to be treated as diffuse. By default, the **Diffuse Fraction** is set to 1, indicating that all of the radiation is diffuse. A diffuse fraction equal to 0 indicates purely specular reflected radiation. A diffuse fraction between 0 and 1 will result in partially diffuse and partially specular reflected energy. See Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide for more details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will also be required to specify the internal emissivity in the **Thermal** tab of the **Wall** dialog box (*Figure 14.17* (p. 727)). For gray-radiation DO models, enter the appropriate value for **Internal Emissivity**. (The default value is 1.) The value that you specify will be applied to the diffuse component only. For non-gray DO models, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box. (The default value in each band is 1.) Alternatively, you can specify the internal emissivity using a boundary condition parameter (see *Creating a New Parameter* (p. 211)).

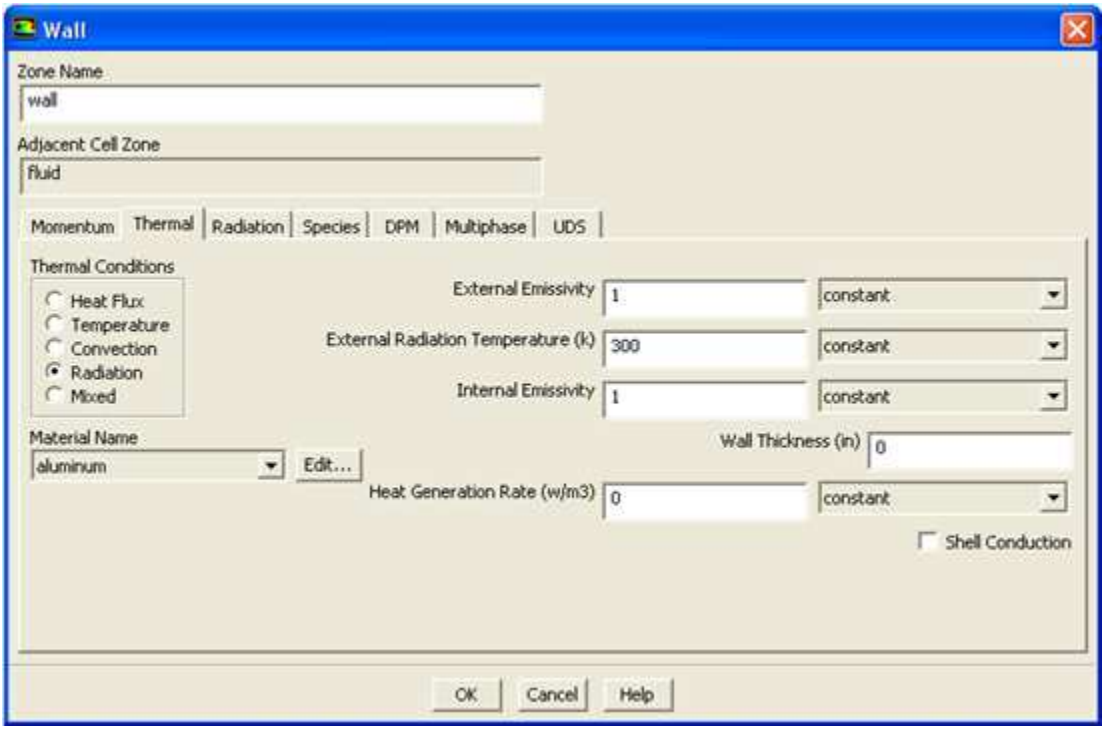
Figure 14.17 The Wall Dialog Box Showing Internal Emissivity Thermal Conditions for an Opaque Wall



You can also specify the external emissivity and external radiation temperature for a semi-transparent wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (*Figure 14.18* (p. 728)). Alternatively, you can specify a UDF for these parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.18 The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions



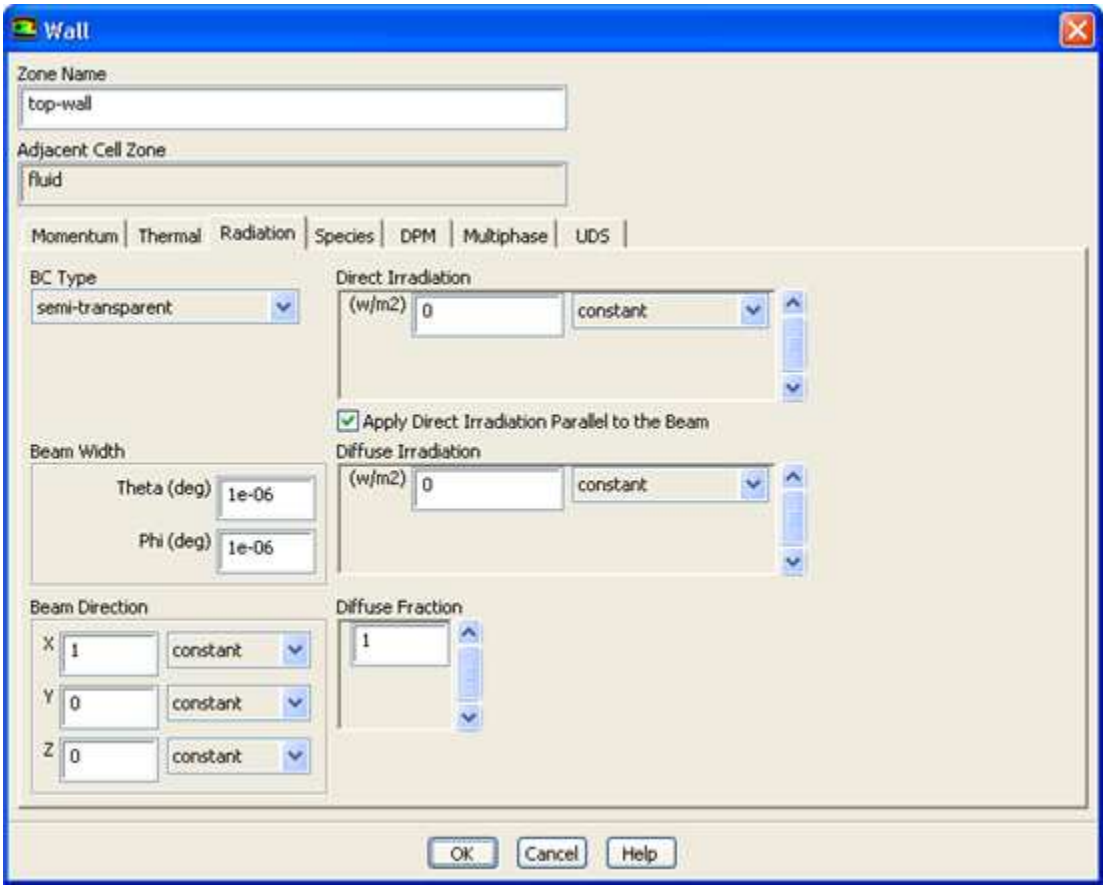
For more information on boundary condition treatment at opaque walls, see Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide.

14.3.7.3.2. Semi-Transparent Walls

To define radiation for an exterior semi-transparent wall, click the **Radiation** tab in the **Wall** dialog box and then select **semi-transparent** in the **BC Type** drop-down list (*Figure 14.19* (p. 729)). The dialog box will expand to display the semi-transparent wall inputs needed to define an external irradiation flux (*Figure 14.19* (p. 729)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.19 The Wall Dialog Box for a Semi-Transparent Wall Boundary



Then perform the following steps:

1. Specify the value of the irradiation flux (in W/m ^{2}) under **Direct** or **Diffuse Irradiation**. If the non-gray DO model is being used, a constant **Direct** or **Diffuse Irradiation** can be specified for each band.

   Important

   Note that the external diffuse irradiation specified when using **Radiation** or **Mixed** thermal conditions (selected in the **Thermal** tab), or **Diffuse Irradiation** (in the **Radiation** tab) is always distributed hemi-spherically after transmission through semi-transparent walls (i.e. independent of whether the external semi-transparent boundary wall is defined as a diffusely or specularly reflecting type).

2. **Apply Direct Irradiation Parallel to the Beam** is the default means of specifying the scale of irradiation flux. When enabled, ANSYS FLUENT assumes that the value of **Direct Irradiation** that you specify is the irradiation flux parallel to the **Beam Direction**. When deselected, ANSYS FLUENT instead assumes that the value specified is the flux parallel to the face normals and will calculate the resulting beam parallel flux for every face. See Figure 5.12: "DO Irradiation on External Semi-Transparent Wall" in Semi-Transparent Exterior Walls in the Theory Guide for details.

3. Define the **Beam Width** by specifying the beam **Theta** and **Phi** extents. Beam width is specified as the solid angle over which the irradiation is distributed. The default value for beam width is $1e^{-}6$ which is suitable for collimated beam radiation. A beam width less than this is likely to result in zero irradiation flux.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the (**X,Y,Z**) vector that defines the **Beam Direction**. The beam direction is defined as the vector of the centroid of the solid angle (beam width). You can specify the **Beam Direction** as a constant, a profile or a UDF. This is especially useful in applications where the shape of the radiative source is circular or cylindrical (or non-linear). For information about boundary profiles, see *Reading and Writing Profile Files* (p. 70).

   Note that the actual direction of the beam of radiation that enters the domain will be further influenced by the solid angles available from the number of divisions set up; the effective direction will be the direction vector of the solid angle that the incoming beam falls into. Finally, any non-zero diffuse fraction will act to spread out (hemispherically, proportional to the diffuse fraction) the irradiation that enters the domain.

   For a UDF example that specifies the beam direction, see Example 5 - Beam Direction Profile at Semi-Transparent Walls in the UDF Manual.

5. Specify the fraction of the irradiation that is to be treated as diffuse as a real number between 0 and 1. By default, the **Diffuse Fraction** is set to 1, indicating that all of the irradiation is diffuse. A diffuse fraction of 0 treats the radiation as purely specular. If you specify a value between 0 and 1, the radiation is treated as partially diffuse and partially specular. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band. See Diffuse Semi-Transparent Walls in the Theory Guide for details.

Important

- Note that the refractive index of the external medium is assumed to be 1.
- If **Heat Flux** conditions are specified in the **Thermal** tab of the **Wall** dialog box, the specified heat flux is considered to be only the conduction and convection portion of the boundary flux. The given irradiation specifies the incoming exterior radiative flux; the radiative flux transmitted from the domain interior to the outside is computed as a part of the calculation by ANSYS FLUENT. Internal emissivity is ignored for semi-transparent surfaces.
- Note that when a boundary wall is made semi-transparent ANSYS FLUENT calculates the amount of radiation leaving as well as entering the domain. If you do not provide a source of irradiation or a radiating thermal condition (e.g. **Mixed** or **Radiation**) then you are effectively radiating to a temperature of 0 K and it is highly likely you may observe temperatures in your model that are lower than expected. Ensure that the external (incoming) radiant conditions give good account of the surroundings.

You can also specify the external emissivity and external radiation temperature for a semi-transparent wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (*Figure 14.18* (p. 728)). Alternatively, you can specify a user-defined function (UDF) for these parameters. For more information, see DEFINE_PROPERTY UDFs in the UDF Manual.

For a detailed description of boundary condition treatment at semi-transparent walls, see Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide.

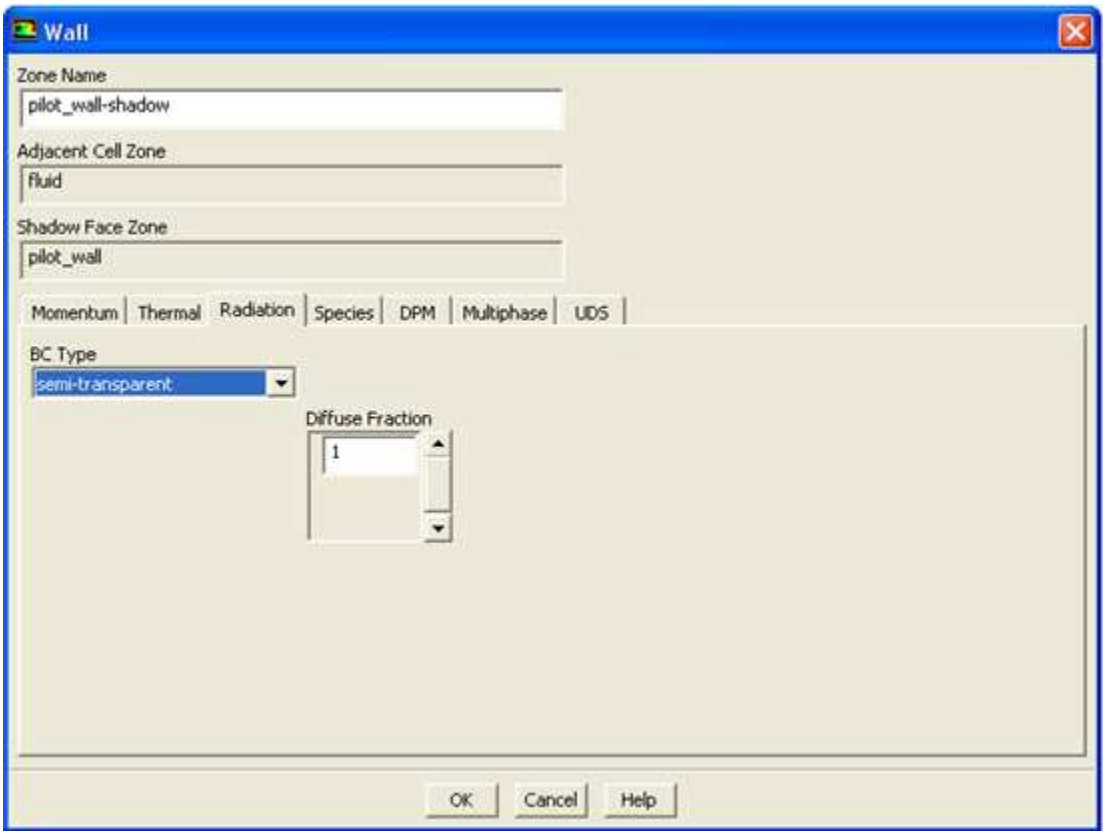
To define radiation for an interior (two-sided) semi-transparent wall, in the **Wall** dialog box click the **Radiation** tab and then select **semi-transparent** in the **BC Type** drop-down list (*Figure 14.20* (p. 731)). Then specify the **Diffuse Fraction** as described for the previous case.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that for semi-transparent walls, the internal emissivity defined under thermal conditions is ignored. Emissivity and absorptivity on a semi-transparent surface can only be effected volumetrically in the wall thickness as a consequence of the wall material absorption coefficient. See notes near the end of Discrete Ordinates (DO) Radiation Model Theory in the Theory Guide discussing limitations around working with the wall thickness.

Figure 14.20 The Wall Dialog Box for an Interior Semi-Transparent Wall

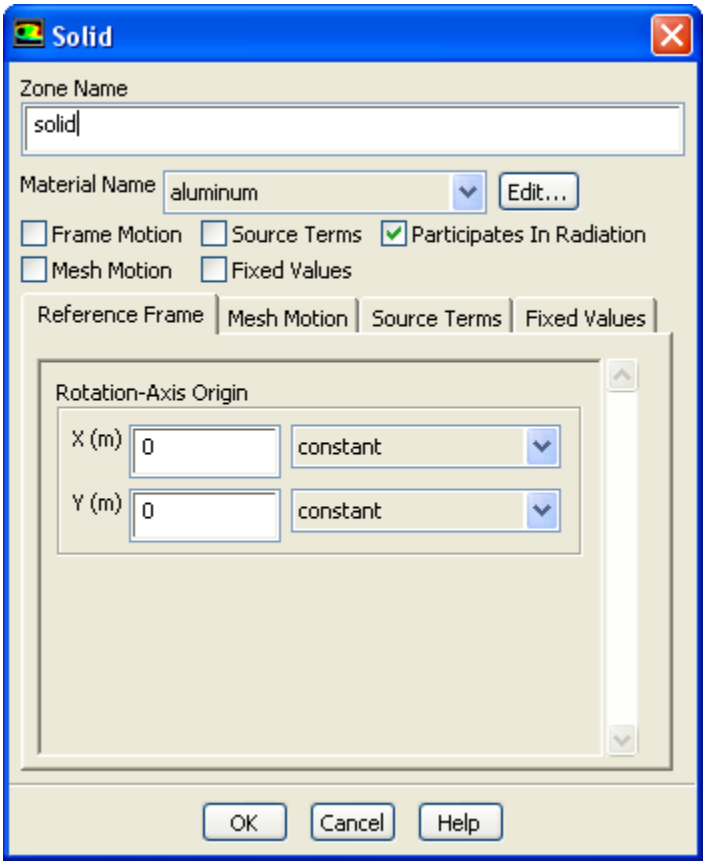


14.3.7.4. Solid Cell Zones Conditions for the DO Model

With the DO model, you can specify whether or not you want to solve for radiation in each cell zone in the domain. By default, the DO equations are solved in all fluid zones, but not in any solid zones. If you want to model semi-transparent media, for example, you can enable radiation in the solid zone(s). To do so, enable the **Participates In Radiation** option in the **Solid** dialog box (*Figure 14.21* (p. 732)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.21 The Solid Dialog Box



Important

In general, you should *not* disable the **Participates In Radiation** option for any fluid zones.

See Solid Semi-Transparent Media in the Theory Guide for more information on solid semi-transparent media.

14.3.7.5. Thermal Boundary Conditions

In general, any well-posed combination of thermal boundary conditions can be used when any of the radiation models is active. The radiation model will be well-posed in combination with fixed temperature walls, conducting walls, and/or walls with set external heat transfer boundary conditions (*Thermal Boundary Conditions at Walls* (p. 309)). You can also use any of the radiation models with heat flux boundary conditions defined at walls, in which case the heat flux you define will be treated as the sum of the convective and radiative heat fluxes. The exception to this is the case of semi-transparent walls for the DO model. Here, ANSYS FLUENT allows you to specify the convective and radiative portions of the heat flux separately.

14.3.8. Solution Strategies for Radiation Modeling

For the P-1, DTRM, S2S, and the DO radiation models, there are several parameters that control the radiation calculation. You can use the default solution parameters for most problems, or you can modify these parameters to control the convergence and accuracy of the solution. Iteration parameters that are unique for a particular radiation model are specified in the **Radiation Model** dialog box (e.g., **Energy Iterations per Radiation Iteration**). **Spatial Discretization** (Discretization in the Theory Guide) and **Under-Relaxation** (Under-Relaxation of Variables in the Theory Guide) are specified in the **Solution Methods** and **Solution**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls task pages, respectively. The **Convergence Criterion** (*Modifying Convergence Criteria* (p. 1276)) is set in the **Residual Monitors** dialog box.

There are no solution parameters to be set for the Rosseland model, since it impacts the solution only through the energy equation.

Important

If radiation is the only model being solved in ANSYS FLUENT, and all other equations are switched off, then the **Energy Iterations per Radiation Iteration** solution parameter that is available for certain radiation models is automatically reset to 1.

14.3.8.1. P-1 Model Solution Parameters

For the P-1 radiation model, you can control the convergence criterion and under-relaxation factor. You should also pay attention to the optical thickness, as described below.

The default convergence criterion for the P-1 model is 10^{-6}, the same as that for the energy equation, since the two are closely linked. See *Monitoring Residuals* (p. 1269) for details about convergence criteria. You can set the **Convergence Criterion** for **p1** in the **Residual Monitors** dialog box.

Monitors → **Residuals** → **Edit...**

The under-relaxation factor for the P-1 model is set with those for other variables, as described in *Setting Under-Relaxation Factors* (p. 1217). Note that since the equation for the radiation temperature (Equation 5–21 in the Theory Guide) is a relatively stable scalar transport equation, in most cases you can safely use large values of under-relaxation (0.9–1.0).

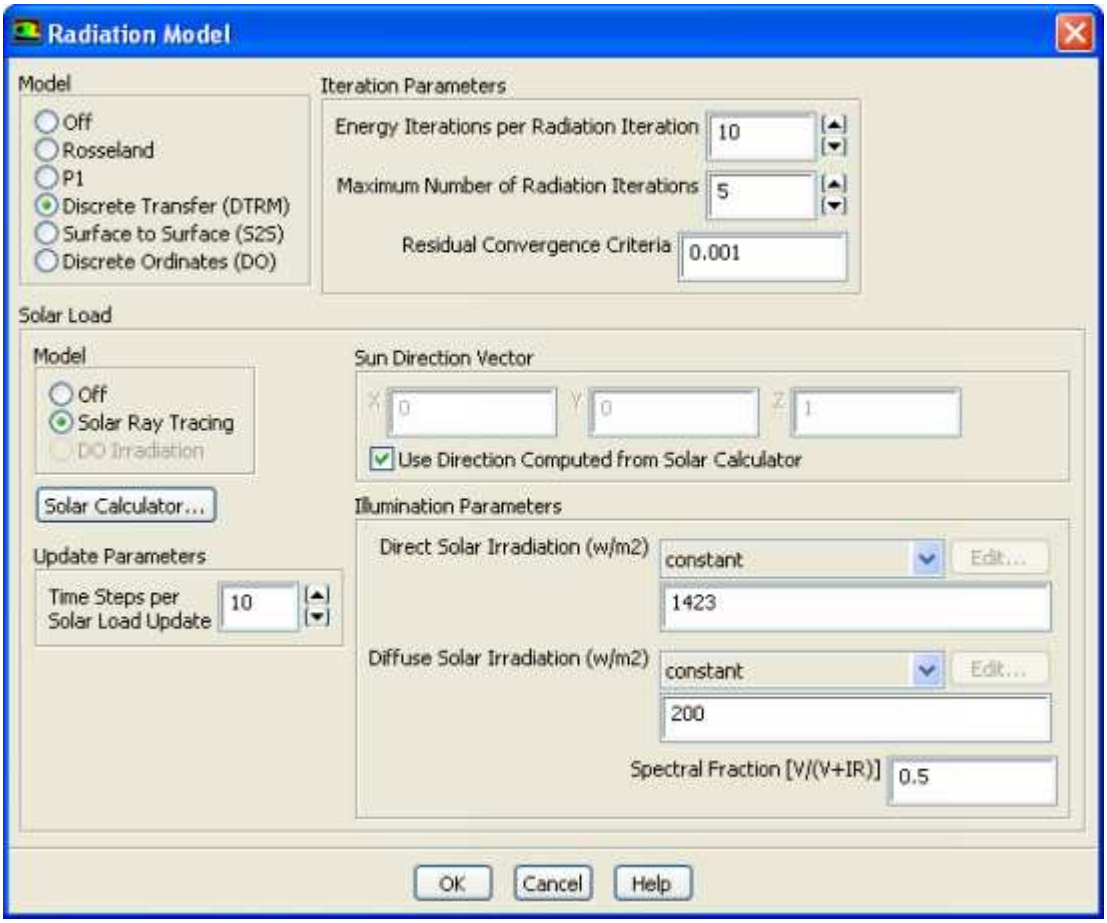
For optimal convergence with the P-1 model, the optical thickness $(a+\sigma_s)L$ must be between 0.01 and 10 (preferably not larger than 5). Smaller optical thicknesses are typical for very small enclosures (characteristic size of the order of 1 cm), but for such problems you can safely increase the absorption coefficient to a value for which $(a+\sigma_s)L=0.01$. Increasing the absorption coefficient will not change the physics of the problem because the difference in the level of transparency of a medium with optical thickness = 0.01 and one with optical thickness <0.01 is indistinguishable within the accuracy level of the computation.

14.3.8.2. DTRM Solution Parameters

When the DTRM is active, ANSYS FLUENT updates the radiation field during the calculation and computes the resulting energy sources and heat fluxes via the ray-tracing technique described in Ray Tracing in the Theory Guide. ANSYS FLUENT provides several solution parameters appear in the expanded portion of the **Radiation Model** dialog box (*Figure 14.22* (p. 734)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.22 The Radiation Model Dialog Box (DTRM)



You can control the maximum number of iterations of the radiation calculation during each global iteration by changing the **Maximum Number of Radiation Iterations**. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

The **Residual Convergence Criteria** parameter (0.001 by default) determines when the radiation intensity update is converged. It is defined as the maximum normalized change in the surface intensity from one DTRM radiation iteration to the next (see *Equation 14–8* (p. 736)).

You can also control the frequency with which the radiation field is updated as the continuous phase solution proceeds. The **Energy Iterations per Radiation Iteration** parameter is set to 10 by default. This means that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.

14.3.8.3. S2S Solution Parameters

For the S2S model, as for the DTRM, you can control the frequency with which the radiosity is updated as the continuous-phase solution proceeds. See the previous description of **Energy Iterations per Radiation Iteration**.

If you are using the pressure-based solver and you first solve the flow equations with the energy equation turned off, you should reduce the **Energy Iterations per Radiation Iteration** from 10 to 1 or 2. This will ensure the convergence of the radiosity. If the default value of 10 is kept in this case, it is possible that the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

flow and energy residuals may converge and the solution will terminate before the radiosity is converged. See *Residual Reporting for the S2S Model* (p. 736) for more information about residuals for the S2S model.

You can control the maximum number of iterations of the radiation calculation during each global iteration by changing the **Maximum Number of Radiation Iterations**. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

The **Residual Convergence Criteria** (0.001 by default) determines when the radiosity update is converged. It is defined as the maximum normalized change in the radiosity from one S2S radiation iteration to the next (see *Equation 14–9* (p. 736)).

14.3.8.4. DO Solution Parameters

For the discrete ordinates model, as for the DTRM, you can control the frequency with which the surface intensity is updated as the continuous phase solution proceeds. See the description of **Energy Iterations per Radiation Iteration** for the DTRM, as mentioned in the previous sections.

For most problems, the default under-relaxation of 1.0 for the DO equations is adequate. For problems with large optical thicknesses ($aL > 10$), you may experience slow convergence or solution oscillation. For such cases, under-relaxing the energy and DO equations is useful. Under-relaxation factors between 0.9 and 1.0 are recommended for both equations.

14.3.8.5. Running the Calculation

Once the radiation problem has been set up, you can proceed as usual with the calculation. Note that while the P-1 and DO models will solve additional transport equations and report residuals, the DTRM and the Rosseland and S2S models will not (since they impact the solution only through the energy equation). Residuals for the DTRM and S2S model radiation iterations are reported by ANSYS FLUENT every time such an iteration is performed, as described below.

14.3.8.5.1. Residual Reporting for the P-1 Model

The residual for radiation as calculated by the P-1 model is updated after each iteration and reported with the residuals for all other variables. ANSYS FLUENT reports the normalized P-1 radiation residual as defined in *Monitoring Residuals* (p. 1269) for the other transport equations.

14.3.8.5.2. Residual Reporting for the DO Model

After each DO iteration, the DO model reports a composite normalized residual for all the DO transport equations. The definition of the residuals is similar to that for the other transport equations (see *Monitoring Residuals* (p. 1269)).

14.3.8.5.3. Residual Reporting for the DTRM

ANSYS FLUENT does not include a DTRM residual in its usual residual report that is issued after each iteration. The effect of radiation on the solution can be gathered, instead, via its impact on the energy field and the energy residual. However, each time a DTRM iteration is performed, ANSYS FLUENT will print out the normalized radiation error for each DTRM radiation iteration. The normalized radiation error is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$E = \frac{\sum_{all\ radiating\ surfaces}\left(I_{new} - I_{old}\right)}{N\left(\sigma T^4/\pi\right)} \tag{14–8}$$

where the error E is the maximum change in the intensity (I) at the current radiation iteration, normalized by the maximum surface emissive power, and N is the total number of radiating surfaces. Note that the default radiation convergence criterion, as noted in *DTRM Solution Parameters* (p. 733), defines the radiation calculation to be converged when E decreases to 10^{-3} or less.

14.3.8.5.4. Residual Reporting for the S2S Model

ANSYS FLUENT does not include an S2S residual in its usual residual report that is issued after each iteration. The effect of radiation on the solution can be gathered, instead, via its impact on the energy field and the energy residual. However, each time an S2S iteration is performed, ANSYS FLUENT will print out the normalized radiation error for each S2S radiation iteration. The normalized radiation error is defined as

$$E = \frac{\sum_{all\ \ radiating\ \ surface\ \ clusters}\left(J_{new} - J_{old}\right)}{N\sigma T^4} \tag{14–9}$$

where the error E is the maximum change in the radiosity (J) at the current radiation iteration, normalized by the maximum surface emissive power, and N is the total number of radiating surface clusters. Note that the default radiation convergence criterion, as noted in *DTRM Solution Parameters* (p. 733), defines the radiation calculation to be converged when E decreases to 10^{-3} or less.

14.3.8.5.5. Disabling the Update of the Radiation Fluxes

Sometimes, you may wish to set up your ANSYS FLUENT model with the radiation model active and then disable the radiation calculation during the initial calculation phase. For the P-1 and DO models, you can turn off the radiation calculation temporarily by deselecting **P1** or **Discrete Ordinates** in the **Equations** list, which is accessed via the **Solution Controls** task page. For the DTRM and the S2S model, there is no item in the **Equations**. You can instead set a very large number for **Energy Iterations per Radiation Iteration** in the **Iteration Parameters** group box of the **Radiation Model** dialog box.

If you turn off the radiation calculation, ANSYS FLUENT will skip the update of the radiation field during subsequent iterations, but will leave in place the influence of the current radiation field on energy sources due to absorption, wall heat fluxes, etc. Turning the radiation calculation off in this way can thus be used to initiate your modeling work with the radiation model inactive and/or to focus the computational effort on the other equations if the radiation model is relatively well converged.

14.3.9. Postprocessing Radiation Quantities

14.3.9.1. Available Variables for Postprocessing

ANSYS FLUENT provides radiation quantities that you can use in postprocessing when your model includes the solution of radiative heat transfer. You can generate graphical plots or alphanumeric reports of the following variables/functions:

In the **Radiation...** category:

- **Incident Radiation** (P-1 and DO models)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Incident Radiation (Band n)** (non-gray P-1 and non-gray DO models)
- **Absorption Coefficient** (DTRM, P-1, DO, and Rosseland models)
- **Scattering Coefficient** (P-1, DO, and Rosseland models)
- **Refractive Index** (P–1, DO, and Rosseland models)
- **Radiation Temperature** (P-1 and DO models)
- **Surface Cluster ID** (S2S model)

In the **Wall Fluxes...** category:

- **Radiation Heat Flux** (all radiation models)
- **Surface Incident Radiation** (S2S, DTRM, and DO models)
- **Absorbed Radiation Flux** (DO model, semi-transparent wall)
- **Reflected Radiation Flux** (DO model, semi-transparent wall)
- **Transmitted Radiation Flux** (DO model, semi-transparent wall)
- **Beam Irradiation Flux** (DO model, semi-transparent wall)

See *Field Function Definitions* (p. 1529) for definitions of these postprocessing variables. Note that in addition, incident radiation, transmitted, reflected and absorbed radiation flux are also available on a per-band basis for the non-gray DO model.

Important

- The sign convention on the radiative heat flux is such that the heat flux from the wall surface is a positive quantity.
- It is possible to export heat flux data on wall zones (including radiation) to a generic file that you can examine or use in an external program. See *Exporting Heat Flux Data* (p. 696) for details.
- Take care not to confuse **Incident Radiation** and **Surface Incident Radiation**. **Incident Radiation** is a volumetric quantity giving the total radiant load passing through the cell (in all directions), whereas **Surface Incident Radiation** is the total radiant load hitting the surface (which will subsequently be absorbed, transmitted and reflected). There is no direct means to report how much radiation has been absorbed/emitted/scattered in cells.

14.3.9.2. Reporting Radiative Heat Transfer Through Boundaries

You can use the **Flux Reports** dialog box to compute the radiative heat transfer through each boundary of the domain, or to sum the radiative heat transfer through all boundaries.

Reports → **Fluxes** → **Set Up...**

See *Fluxes Through Boundaries* (p. 1510) for details about generating flux reports.

14.3.9.3. Overall Heat Balances When Using the DTRM

The DTRM yields a global heat balance and a balance of radiant heat fluxes only in the limit of a sufficient number of rays. In any given calculation, therefore, if the number of rays is insufficient you may find that the radiant fluxes do not obey a strict balance. Such imbalances are the inevitable consequence of the discrete ray tracing procedure and can be minimized by selecting a larger number of rays from each wall boundary.

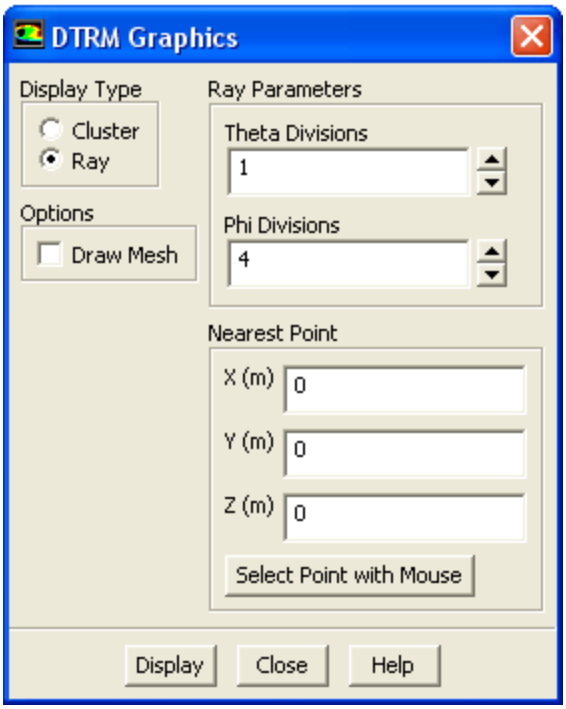
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.9.4. Displaying Rays and Clusters for the DTRM

When you use the DTRM, ANSYS FLUENT allows you to display surface or volume clusters, as well as the rays that emanate from a particular surface cluster. You can use the **DTRM Graphics** dialog box (*Figure 14.23* (p. 738)) for all of these displays.

Display → **DTRM Graphics...**

Figure 14.23 The DTRM Graphics Dialog Box



14.3.9.4.1. Displaying Clusters

To view clusters, select **Cluster** under **Display Type** and then select either **Surface** or **Volume** under **Cluster Type**.

To display all of the surface or volume clusters, select the **Display All Clusters** option under **Cluster Selection** and click the **Display** button.

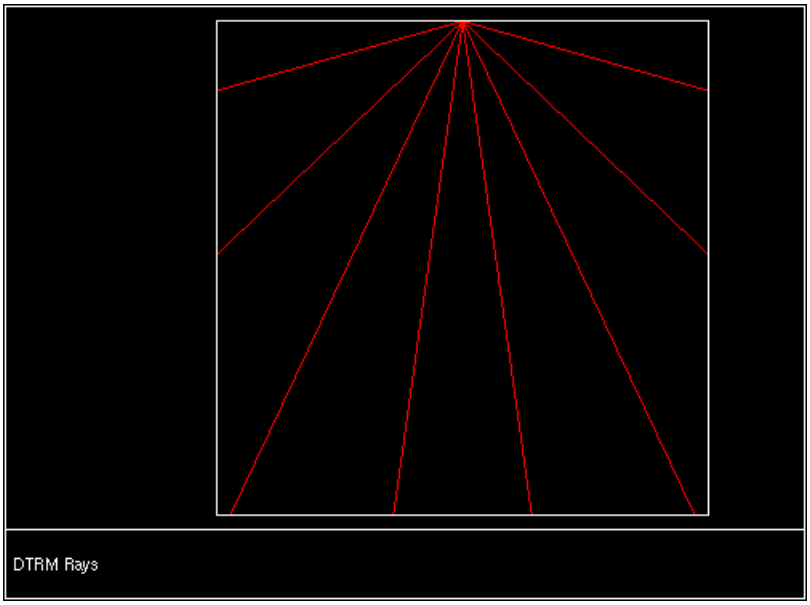
To display only the cluster (surface or volume) nearest to a specified point, deselect the **Display All Clusters** option and specify the coordinates under **Nearest Point**. You may also use the mouse to choose the nearest point. Click on the **Select Point With Mouse** button and then right-click on a point in the graphics window.

14.3.9.4.2. Displaying Rays

To display the rays emanating from the surface cluster nearest to the specified point, select **Ray** under **Display Type**. Set the appropriate values for **Theta** and **Phi Divisions** under **Ray Parameters** (see *Setting Up the DTRM* (p. 707) for details), and then click the **Display** button. *Figure 14.24* (p. 739) shows a ray plot for a simple 2D geometry.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.24 Ray Display



14.3.9.4.3. Including the Mesh in the Display

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your ray or cluster display as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with displaying the rays. This is accomplished by enabling the **Draw Mesh** option in the **DTRM Graphics** dialog box. The **Mesh Display** dialog box will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the **DTRM Graphics** dialog box, the mesh display, as defined in the **Mesh Display** dialog box, will be included in the ray or cluster display.

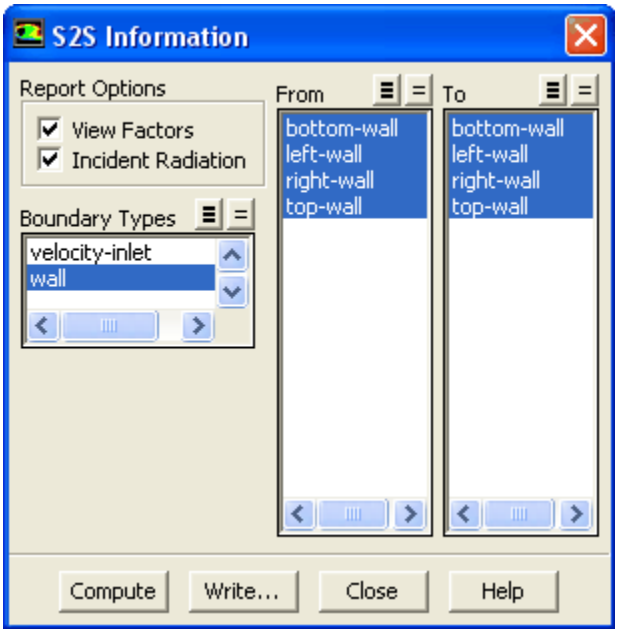
14.3.9.5. Reporting Radiation in the S2S Model

When you use the S2S model, ANSYS FLUENT allows you to view the values of the view factor and radiation emitted from one zone to any other zone. You can use the **S2S Information** dialog box (*Figure 14.25* (p. 740)) to generate a report of these values in the console or as a separate file.

Report → **S2S Information...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.25 The S2S Information Dialog Box



The steps for generating the report are as follows:

1. Specify the values in which you are interested by selecting **View Factors** and/or **Incident Radiation**.
2. Choose the zones for which you would like data by selecting them in the lists under **From** and **To** (at least one zone must be selected under each list). To select all of the zones of a particular type, click on that category in the list under **Boundary Types**.
3. Specify how you would like to present the data. To report the values in the console, click the **Compute** button. To write the data as an S2S Info File (`.sif` format), click the **Write...** button and enter a file name in *The **Select File** Dialog Box* (p. 33).

The following is an example of how the data is presented:

```
S2S Information
From wall1 to:
                        Viewfactor        Incident Radiation
            wall1           0.0000                    0.0000
            wall2           0.2929                171387.7813
            wall3           0.2929                155305.7969
            wall4           0.4142                 29055.9023

From wall2 to:
                        Viewfactor        Incident Radiation
            wall1           0.2929                306451.9688
            wall2           0.0000                    0.0000
            wall3           0.4142                214195.0938
            wall4           0.2929                 19153.2715
```

Note that the header listed above (`S2S Information`) is not displayed in the console.

14.3.10. Solar Load Model

ANSYS FLUENT provides a solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain. Two options are available for the model: solar ray tracing and DO irradiation. The ray tracing approach is a highly efficient and practical means of applying solar loads as heat sources in the energy equations. In cases where you want to use the discrete ordinates (DO) model to cal-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

culate radiation effects within the domain, an option is available to supply outside beam direction and intensity parameters directly to the DO model. The solar load model includes a solar calculator utility that can be used to construct the sun's location in the sky for a given time-of-day, date, and position. Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.

14.3.10.1. Introduction

Typical applications that are well-suited for solar load simulations include the following:

- automotive climate control (ACC) applications
- human comfort modeling applications in buildings

The effects of solar loading are needed in many ACC applications, where the temperature, humidity, and velocity fields around passengers (and drivers) are desired. ACC systems are tested for their capacity to cool down passenger compartments after they have been "soaked" in intense solar radiation. ANSYS FLUENT's solar load model will enable you to simulate solar loading effects and predict the time it will take to reasonably cool down the cabin of a car that has been exposed to solar radiation, as well as predict the time interval needed to lower the temperature in specified points and areas within the domain.

In the analysis of buildings, solar loading provides a significant burden on the cooling requirement in warm climates, particularly where architects want to use the aesthetics of glazed facades. Even in cooler climates, solar loading can provide a burden during warmer seasons where modern buildings are well insulated against thermal loss during winter months. As well as providing an engineer with a practical tool for determining the solar heating effect inside a building, ANSYS FLUENT's solar load model will allow the solar transmission through all glazed surfaces to be determined over the course of a day, allowing important decisions to be made before undertaking any flow studies. ANSYS FLUENT's solar load model also allows you to simulate porous blinds, which can transmit a portion of the solar radiation while also allowing fluid flow.

14.3.10.2. Solar Ray Tracing

The solar load model's ray tracing algorithm can be used to predict the direct illumination energy source that results from incident solar radiation. It takes a beam that is modeled using the sun position vector and illumination parameters, applies it to any or all wall, porous jump, and inlet/outlet boundary zones that you specify, performs a face-by-face shading analysis to determine well-defined shadows on all boundary faces and interior walls, and computes the heat flux on the boundary faces that results from the incident radiation.

Important

The solar ray tracing model includes only boundary zones that are adjacent to fluid zones in the ray tracing calculation. In other words, boundary zones that are attached to solid zones are ignored.

The resulting heat flux that is computed by the solar ray tracing algorithm is coupled to the ANSYS FLUENT calculation via a source term in the energy equation. The heat sources are added directly to computational cells bordering each face and are assigned to adjacent cells in the following order: shell conduction cells, solid cells, and fluid cells. Heat sources are assigned to one of these types of adjacent cells, only. You can choose to override this order and include adjacent fluid cells in the solar load calculation by issuing a command in the text user interface (see *Text Interface-Only Commands* (p. 759) for details). Note that the sun position vector and solar intensity can be entered either directly by you or computed from the solar calculator. Direct and diffuse irradiation parameters can also be specified using a user-defined function (UDF) and hooked to ANSYS FLUENT in the **Radiation Model** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The solar ray tracing option allows you to include the effects of direct solar illumination as well as diffuse solar radiation in your ANSYS FLUENT model. A two-band spectral model is used for direct solar illumination and accounts for separate material properties in the visible and infrared bands. A single-band hemispherical-averaged spectral model is used for diffuse radiation. Opaque materials are characterized in terms of two-band absorptivities. A semi-transparent material requires specification of absorptivity and transmissivity. Values that you specify for transmissivity and absorptivity are defined for normal incident rays. ANSYS FLUENT recomputes/interpolates these values for the given angle of incidence.

The solar ray tracing algorithm also accounts for internal scattered and diffusive loading. The reflected component of direct solar irradiation is tracked. A fraction of this radiative heat flux, called internally scattered energy is applied to all the surfaces participating in the solar load calculation, weighted by area. The internally scattered energy depends on the scattering fraction which is specified in the TUI, and whose default value is 1. Depending on the reflectivity of the primary surface, the scattering fraction can be responsible for the inclusion (or exclusion) of a large amount of radiation within the rest of the domain.

Also included as internally scattered energy is the contribution of the transmitted component of diffuse solar irradiation (which enters a domain through semi-transparent walls depending upon the hemispherical transmissivity). The total value of internally scattered energy is reported to the ANSYS FLUENT console. The ambient flux is obtained by dividing the internally scattered energy by the total surface area of the faces participating in the solar load calculation.

Note that **Solar Ray Tracing** is *not* a participating radiation model. It does not deal with emission from surfaces, and the reflecting component of the primary incident load is distributed uniformly across all surfaces rather than being local to the surfaces reflected to. If surface emission is an important factor in your case then you can consider implementing a radiation model (e.g., P-1) in conjunction with **Solar Ray Tracing**.

14.3.10.2.1. Shading Algorithm

The shading calculation that is used for solar ray tracing is a straightforward application of vector geometry. A ray is traced from the centroid of a test face in the direction of the sun. Every other face is checked to determine if the ray intersects the candidate face and if the candidate face is in front of the test face. If both conditions are met, then an opaque face completely shades the test face. A semi-transparent face attenuates the incident energy.

A Barycentric coordinate formulation is used to construct triangle-ray intersections. A quadrilateral ray intersection method is used to handle the case when model surfaces contain quadrilaterals. A quad-tree preprocessing step is applied to reduce the ray tracing algorithm complexity that can lead to long runtime for 10^4 faces and greater. The quad-tree refinement factor can be modified in the text interface. The default value of this parameter is 7 which is sufficient to cover the entire spectrum of mesh sizes between one cell and five million cells. If the mesh is greater than five million cells, an increase in this parameter would reduce the CPU time needed to compute the solar loads.

14.3.10.2.2. Glazing Materials

Incident solar radiation can be applied to glass and plastic glazing materials of various types at wall boundaries, and the effects of coated glazings modeled using the solar ray tracing algorithm. To model solar optical properties, you will need to specify the transmissivity and reflectivity of the material in the **Wall** boundary conditions dialog box. You can obtain these values from the glass (or plastic) manufacturer or use data from another source (e.g., ASHRAE Handbook).

Glazing optical properties are dependent on incident angle, and the variation is significant for an incident angle greater than 40 degrees. As the incident angle increases from zero, transmissivity decreases, reflectivity increases, and absorptivity increases initially due to lengthened optical path, and then decreases as more

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

incident radiation is reflected. The shape of the property curve varies with glass type and thickness. This difference is more pronounced for coated glass or for a multiple-pane glazing system. It cannot be assumed that all glazing systems have a universal angular dependence.

For coated glazings, the spectral transmissivity and reflectivity at any incident angle are approximated in the solar load model from the normal angle of incidence [*24*] (p. 2210).

Transmissivity is given by

$$T(\theta,\lambda)=T(0,\lambda)\,Tref(\theta) \tag{14–10}$$

where

$$Tref(\theta)=a0+a1cos(\theta)+a^2cos\left(\theta^2\right)+a3cos\left(\theta^3\right)+a4cos\left(\theta^4\right) \tag{14–11}$$

Reflectivity is given by

$$R(\theta,\lambda)=R(0,\lambda)\,[1-Rref(\theta)]+Rref(\theta) \tag{14–12}$$

where

$$Rref(\theta)=b0+b1cos(\theta)+b^2cos\left(\theta^2\right)+b3cos\left(\theta^3\right)+b4cos\left(\theta^4\right)-Tref(\theta) \tag{14–13}$$

The constants used in *Equation 14–10* (p. 743) and *Equation 14–12* (p. 743) are for coated glazings and are taken from Finlayson and Arasteh. [*24*] (p. 2210). The normal transmissivity and reflectivity, $T(0,\lambda)$ and $R(0,\lambda)$ are specified in the **Wall** boundary conditions dialog box.

14.3.10.2.3. Inputs

The following inputs are required for the solar ray tracing algorithm:

- sun direction vector
- direct solar irradiation
- diffuse solar irradiation
- spectral fraction
- direct and IR absorptivity (opaque wall)
- direct and IR absorptivity and transmissivity (semi-transparent wall and porous jump)
- diffuse hemispherical absorptivity and transmissivity (semi-transparent wall)
- solar transmissivity factor
- quad tree refinement factor
- scattering fraction
- ground reflectivity

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sun direction vector is the direction vector looking to the sun, from which the direct irradiation will be incident. You can enter the vector components (**X**, **Y**, **Z**) and the direct and diffuse solar irradiation fluxes in the **Radiation Model** dialog box, or you can have these parameters derived from the solar calculator. These irradiation fluxes can also be specified using a user-defined function (*User-Defined Functions (UDFs) for Solar Load* (p. 748)). The spectral fraction is the final input in the **Radiation Model** dialog box. This defines the split of visible and infra-red (shortwave and longwave respectively) radiation, specifically the fraction of the direct irradiation flux that is in the visible band. These quantities can also be defined through the text interface.

The scattering fraction defines the amount of non-absorbed radiation that will be distributed (uniformly) across all participating surfaces. This is required as the solar load model does not track the rays beyond the first opaque surface. Therefore, a highly glazed space where incident radiation is likely to be reflected back out will have a low value. Conversely, a predominantly opaque (wall-bounded) space where reflected radiation is likely to be incident upon (and ultimately absorbed by) other opaque surfaces will have a high value. This parameter is defined through the text interface only, taking a default value of 1.0:

`define` → `models` → `radiation` → `solar-parameters` → `scattering-fraction`

The ground reflectivity is used by the solar calculator to compute the background diffuse radiation intensity component contributed to by radiation reflected off the ground. This should be based on typical figures for the surface reflectivity of the outside ground surfaces. By default this is set to 0.2, but can be adjusted through the text-interface:

`define` → `models` → `radiation` → `solar-parameters` → `ground-reflectivity`

The quad-tree-refinement parameter determines the level of detail used by the shading algorithm. By default this is set to 7 which will generally work well, but can lie between 0 and 10. This is defined only through the text interface:

`define` → `models` → `radiation` → `solar-parameters` → `quad-tree-refinement`

Further details on the text interface-only entries is provided later in this section (see Text Interface-Only Commands).

The absorptivity and transmissivity parameters related to a wall or porous jump are entered in the **Wall** (under the **Radiation** tab) or **Porous Jump** boundary condition dialog box, respectively, for the particular boundary zones you wish to participate in solar ray tracing. On flow boundaries you have a solar transmissivity factor to allow you to attenuate the incoming solar flux, e.g. set to 1 for a fully open inlet or set to 0 for a light obscuring louvered inlet.

14.3.10.3. DO Irradiation

The solar load model's discrete ordinates (DO) irradiation option provides you with an easy means of applying a solar load directly to the DO model. Unlike the ray tracing solar load option, the DO irradiation method does not compute heat fluxes and apply them as heat sources to the energy equation. Instead, the irradiation flux is applied directly to semi-transparent walls (which you specify) as a boundary condition, and the radiative heat transfer is derived from the solution of the DO radiative transfer equation.

The following inputs are required for DO irradiation at semi-transparent walls:

- direct irradiation
- diffuse irradiation
- beam direction

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- beam width
- diffuse fraction

In the **Wall** boundary condition dialog box for each semi-transparent wall you want to participate in DO irradiation, you can specify that the beam direction, direct irradiation, and diffuse irradiation be derived from the solar parameters (e.g., solar calculator) which you set (or compute) in the **Radiation Model** dialog box. This is done by checking the **Use Beam Direction from Solar Load Model Settings** and **Use Direct and Diffuse Irradiation from Solar Load Model Settings** boxes. When selected, ANSYS FLUENT sets the beam width (the angle subtended by the sun) to the default value of 0.53 degrees for DO irradiation.

Important

Note that the sign of the beam direction that is needed for the DO model is opposite the sun direction vector that is entered or derived from the solar parameters. The beam direction in the DO model is the direction of external radiation (e.g., radiation coming from the sun), while the sun direction vector in the solar load model points to the sun. Incident radiation and the sun angle always have an opposite sign since they are quantities that are defined from opposite perspectives.

14.3.10.4. Solar Calculator

ANSYS FLUENT provides a solar calculator that can be used to compute solar beam direction and irradiation for a given time, date, and position. These values can be used as inputs to the solar ray tracing algorithm or as semi-transparent wall boundary conditions for discrete ordinates (DO) irradiation.

14.3.10.4.1. Inputs/Outputs

Inputs needed for the solar calculator are:

- global position (latitude, longitude, time zone)
- starting date and time
- mesh orientation
- solar irradiation method
- sunshine factor

Global position consists of latitude, longitude, and time zone (relative to GMT). The time of day for a transient simulation is the starting time plus the flow-time. For mesh orientation, you will need to specify the North and East direction vector in the CFD mesh. The default solar irradiation method is Fair Weather Conditions. Alternatively, you can choose the Theoretical Maximum method. The sunshine factor is simply a linear reduction factor for the computed incident load that allows for cloud cover to be accounted for, if appropriate.

You can specify these inputs in the **Solar Calculator** dialog box that is accessible from the **Radiation Model** dialog box (*Figure 14.29* (p. 752)). Alternatively, you can enter the parameters using text interface commands (*Additional Text Interface Commands* (p. 761)).

The following values are computed by the solar calculator and are displayed in the console whenever the solar calculator is used:

- sun direction vector
- sunshine fraction

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- direct normal solar irradiation at earth's surface
- diffuse solar irradiation - vertical and horizontal surface
- ground reflected (diffuse) solar irradiation - vertical surface

Direct normal solar irradiation is computed using the ASHRAE Fair Weather Conditions method, when this option is selected in the solar calculator. Note: Equation 20 and Table 7 from Chapter 30 of the 2001 ASHRAE Handbook of Fundamentals. The theoretical maximum values for direct normal solar irradiation and diffuse solar irradiation are computed using NREL's Theoretical Maximum method, when this option is selected. In practice, these values are unlikely to be experienced due to atmospheric conditions.

ANSYS FLUENT computes the diffuse solar irradiation components (vertical and horizontal) internally for each face in the domain. When the Theoretical Maximum method is chosen, these diffuse irradiation values provide estimates for the maximum vertical and horizontal surface effects.

14.3.10.4.2. Theory

ANSYS FLUENT provides two options for computing the solar load: Fair Weather Conditions method and Theoretical Maximum method. Although these methods are similar, there is a key difference. The Fair Weather Conditions method imposes greater attenuation on the solar load which is representative of atmospheric conditions that are fair –but not completely clear.

The equation for normal direct irradiation applying the Fair Weather Conditions Method is taken from the ASHRAE Handbook:

$$Edn = \frac{A}{e^{\frac{B}{\sin(\beta)}}} \tag{14–14}$$

where A and B are apparent solar irradiation at air mass $m=0$ and atmospheric extinction coefficient, respectively. These values are based on the earth's surface on a clear day. β is the solar altitude (in degrees) above the horizontal.

The equation for direct normal irradiation that is used for the Theoretical Maximum Method is taken from NREL's Solar Position and Intensity Code (Solpos):

$$Edn = S_{etrn} S_{unprime} \tag{14–15}$$

where S_{etrn} is the top of the atmosphere direct normal solar irradiance and $S_{unprime}$ is the correction factor used to account for reduction in solar load through the atmosphere.

The calculation for the diffuse load in the solar model is based on the approach suggested in the 2001 ASHRAE Fundamental Handbook (Chapter 20, Fenestration). The equation for diffuse solar irradiation on a vertical surface is given by:

$$Ed = CYEdn \tag{14–16}$$

where C is a constant whose values are given in Table 7 from Chapter 30 of the 2001 ASHRAE Handbook of Fundamentals, Y is the ratio of sky diffuse radiation on a vertical surface to that on a horizontal surface (calculated as a function of incident angle), and Edn is the direct normal irradiation at the earth's surface on a clear day.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The equation for diffuse solar irradiation for surfaces other than vertical surfaces is given by:

$$Ed = CEdn\frac{(1+\cos\varepsilon)}{2} \quad \textbf{(14–17)}$$

where ε is the tilt angle of the surface (in degrees) from the horizontal plane.

The equation for ground reflected solar irradiation on a surface is given by:

$$Er = Edn\,(C + \sin\beta)\,\rho_g\frac{(1-\cos\varepsilon)}{2} \quad \textbf{(14–18)}$$

where ρ_g is the ground reflectivity. The total diffuse irradiation on a given surface will be the sum of Ed and Er when the input for diffuse solar radiation is taken from the solar calculator. Otherwise, if the `constant` option is selected in the **Radiation** dialog box, then the total diffuse irradiation will be the same as specified in the dialog box.

14.3.10.4.3. Computation of Load Distribution

In calculating the solar load that will be incident on each surface, it is necessary to distinguish between the calculation of diffuse and direct solar loads. A direct load will be tracked from participating transmissive boundary surfaces and non-participating boundary surfaces, the former provides some opportunity to attenuate the incoming flux by absorption and reflection, while the non-participating surfaces allow the flux to enter without any drop in intensity. The direct load is then tracked through the model space until it is incident on an opaque surface, or it exists through a transmissive or non-participating boundary zone. During its passage, its intensity will be attenuated as it passes through participating semi-transparent internal walls, where some radiation may be absorbed and some may be reflected. The total amount of direct radiation which is reflected at internally facing surfaces will be added to the scattered radiation budget for further use later.

The diffuse load originates at participating transmissive boundary surfaces. It is these surfaces that permit diffuse radiation to enter, irrespective of their orientation relative to the direction vector. For each transmissive surface, some of the incoming diffuse load may be immediately absorbed and/or reflected to the outside. The rest is assumed to be transmitted inside and summed from all of these surfaces to give an initial diffuse budget. Onto this budget is added a fraction of the previously computed scattered radiation from the direct load, the fraction used is defined as an input to the model. This provides the total diffuse load. This is then uniformly distributed across all surfaces which are participating in the solar calculation, irrespective of whether they are opaque or semi-transparent. There is no scope to define local absorptivity for this distribution and no biasing with regards proximity to transmissive surfaces. Note that a non-participating boundary zone will allow direct load to enter the model space but will not provide an incoming quantity of diffuse load.

Note that the solar flux which is externally incident on an opaque surface will be completely disregarded, e.g. solar load on an opaque roof of a model whose internals only are modeled will not be included as a heat gain. Instead, this heat gain should be manually calculated and applied as a thermal condition, typically using a fixed heat flux or a radiation/mixed condition.

14.3.10.5. Using the Solar Load Model

When you want to run a steady-state solution with solar load enabled, you simply set up the solar load model (*Setting Up the Solar Load Model* (p. 748)) and boundary conditions (*Setting Boundary Conditions for Solar Loading* (p. 753)) for your case, and then run the simulation. The solution data file will contain the solar

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fluxes that you can use for postprocessing. For a steady-state solution, the solar loads are computed on initialization. If you want to initially solve a case without solar loading (say, for stability) and then add the effects of solar loading afterward, you will need to enable the solar load model through the text user interface (TUI).

Important

Note that you can compute the solar load at any time once you have set up the model by using the `sol-on-demand` text interface command (see *Additional Text Interface Commands* (p. 761) for details).

When you want to run a transient solar load simulation, the process is the same as for the steady-state case but you will need to specify the additional **Time Steps per Solar Load Update** parameter in the **Radiation Model** dialog box. ANSYS FLUENT will re-compute the sun position and irradiation and update solar loads with this specified frequency.

Important

Note that parallel simulations involving the solar load model are set up and computed using the same steps as in serial.

14.3.10.5.1. User-Defined Functions (UDFs) for Solar Load

You can write a user-defined function (UDF) to specify direct and diffuse solar intensity using the `DEFINE_SOLAR_INTENSITY` macro. See DEFINE_SOLAR_INTENSITY in the UDF Manual for more information. After it is interpreted or compiled, you can hook your intensity UDF for direct or diffuse solar irradiation by selecting `user-defined` in the drop-down lists for these parameters in the **Radiation Model** dialog box. See Step 2 in *Setting Up the Solar Load Model* (p. 748) for details.

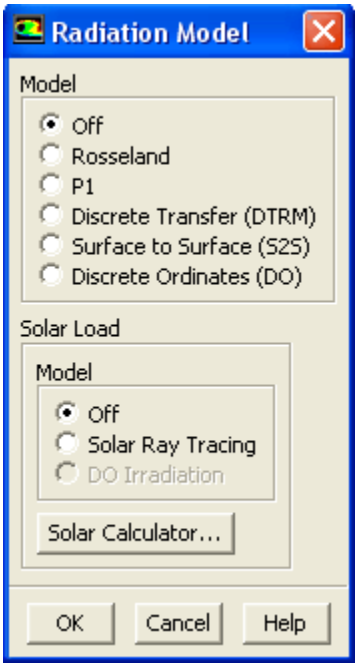
14.3.10.5.2. Setting Up the Solar Load Model

The solar load model is enabled in the **Radiation Model** dialog box (*Figure 14.26* (p. 749)).

Models → **Radiation** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.26 The Radiation Model Dialog Box



Important

Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.

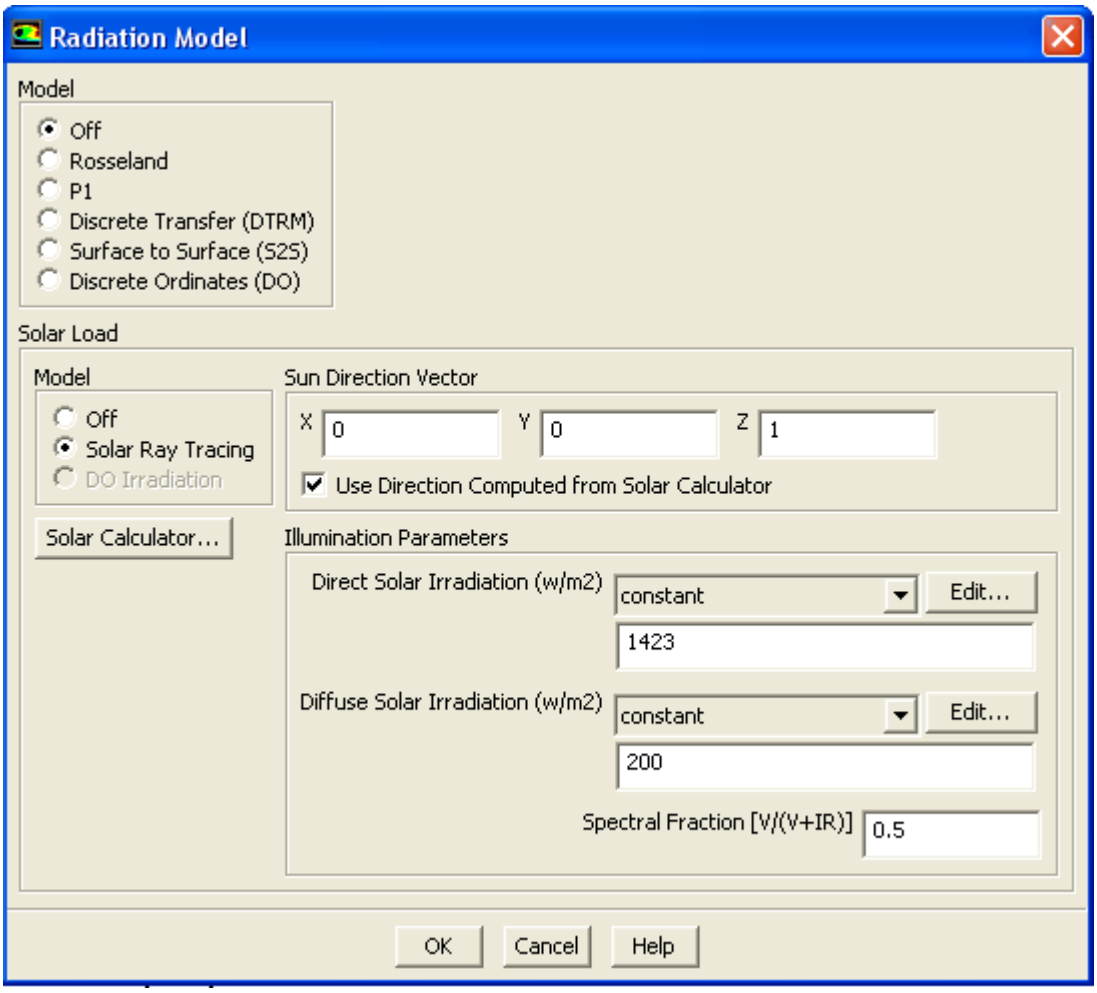
The solar load model has two options: **Solar Ray Tracing** and **DO Irradiation**. **Solar Ray Tracing** can be applied as a standalone solar loading model, or it can be used in conjunction with one of the ANSYS FLUENT radiation models (P1, Rosseland, Discrete Transfer, Surface-to-Surface, Discrete Ordinates). **DO Irradiation** is available only when the **Discrete Ordinates (DO)** radiation model is enabled.

To set up the solar load model, perform the following steps:

1. Enable the solar load model in the **Radiation Model** Dialog Box.
   a. To enable the solar ray tracing algorithm, select **Solar Ray Tracing** under **Solar Load** (*Figure 14.27* (p. 750)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

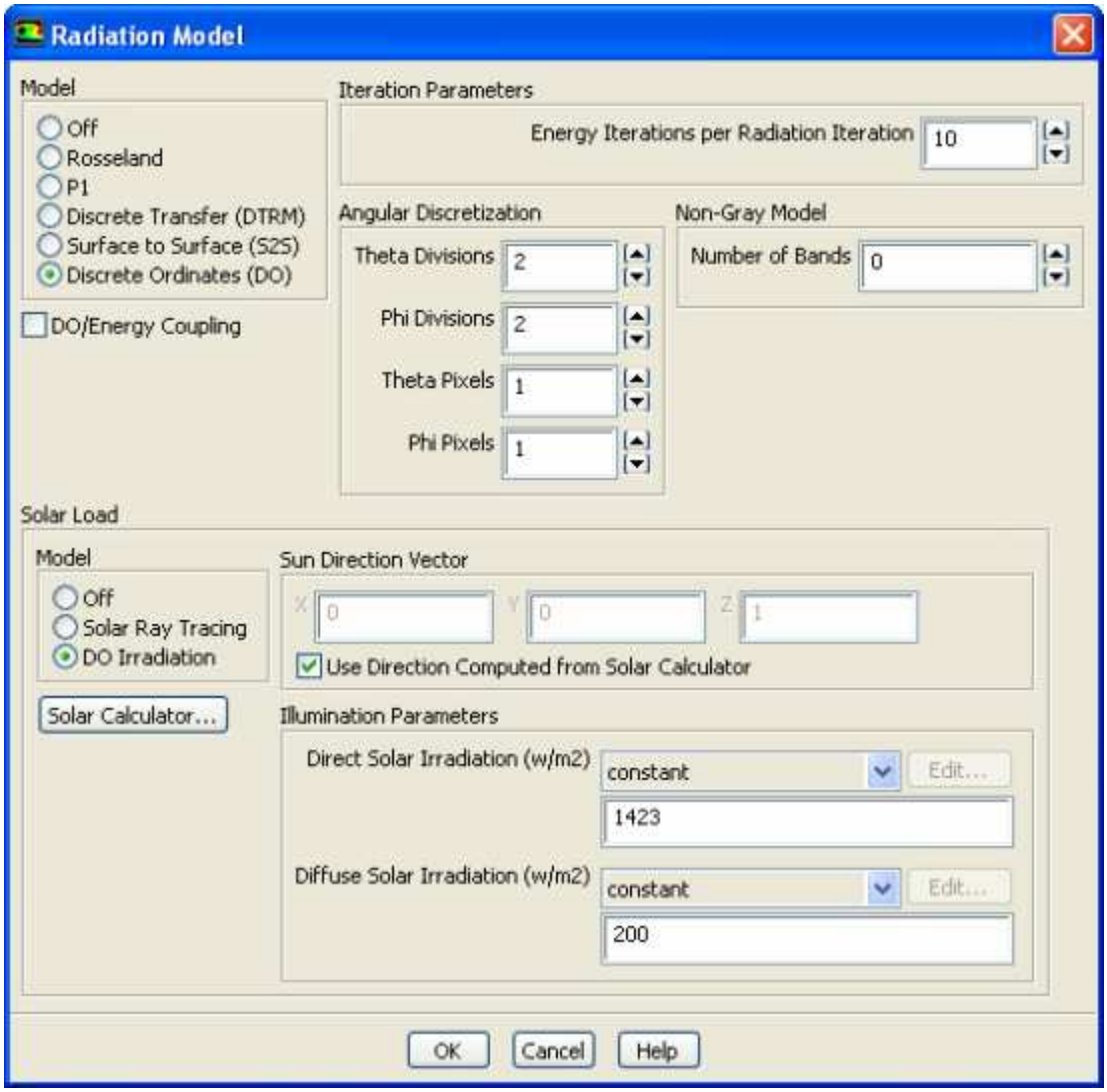
Figure 14.27 The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option)



b. To enable the DO irradiation option, first select **Discrete Ordinates** under **Model**, and then select **DO Irradiation** under **Solar Load** (*Figure 14.28* (p. 751)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.28 The Radiation Model Dialog Box (with Solar Load Model DO Irradiation Option)



2. Define the solar parameters.

   a. Enter values for the **X**, **Y**, and **Z**components of the **Sun Direction Vector**. Alternatively, you can choose to have this vector computed from the solar calculator by enabling the **Use Direction Computed from Solar Calculator** option.

   b. Specify the illumination parameters.

      i. Enter a value for **Direct Solar Irradiation** under **Illumination Parameters**. This parameter is the amount of energy per unit area in W/m^2 due to direct solar irradiation. This value may depend on the time of year and the clearness of the sky. Make your selection in the drop-down list next to **Direct Solar Irradiation** and either enter a **constant** value, have the value computed from the **solar calculator**, or specify it using a **user-defined function**. (For more information on writing solar intensity UDFs, see DEFINE_SOLAR_INTENSITY in the UDF Manual.) For transient simulations, you have the additional option of specifying a time-dependent **piecewise-linear** and **polynomial** profile for direct solar irradiation.

      ii. Enter a value for **Diffuse Solar Irradiation**, which is the amount of energy per unit area in W/m^2 due to diffuse solar irradiation. This value may depend on the time of year, the clearness

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the sky, and also on ground reflectivity. Make your selection in the drop-down list next to **Diffuse Solar Irradiation** and either enter a **constant** value, have the value computed from the **solar calculator**, or specify it using a **user-defined function**. (For more information on writing solar intensity UDFs, see DEFINE_SOLAR_INTENSITY in the UDF Manual.) For transient simulations, you have the additional option of specifying a time-dependent **piecewise-linear** and **polynomial** profile for diffuse solar irradiation.

iii. If you are using the **Solar Ray Tracing** solar load model (*Figure 14.27* (p. 750)), then you will need to enter a value for **Spectral Fraction**. The spectral fraction is the fraction of incident solar radiation in the visible part of the solar radiation spectrum. The spectral fraction is not used for DO irradiation since the DO implementation is intended only for a single band.

Spectral Fraction =

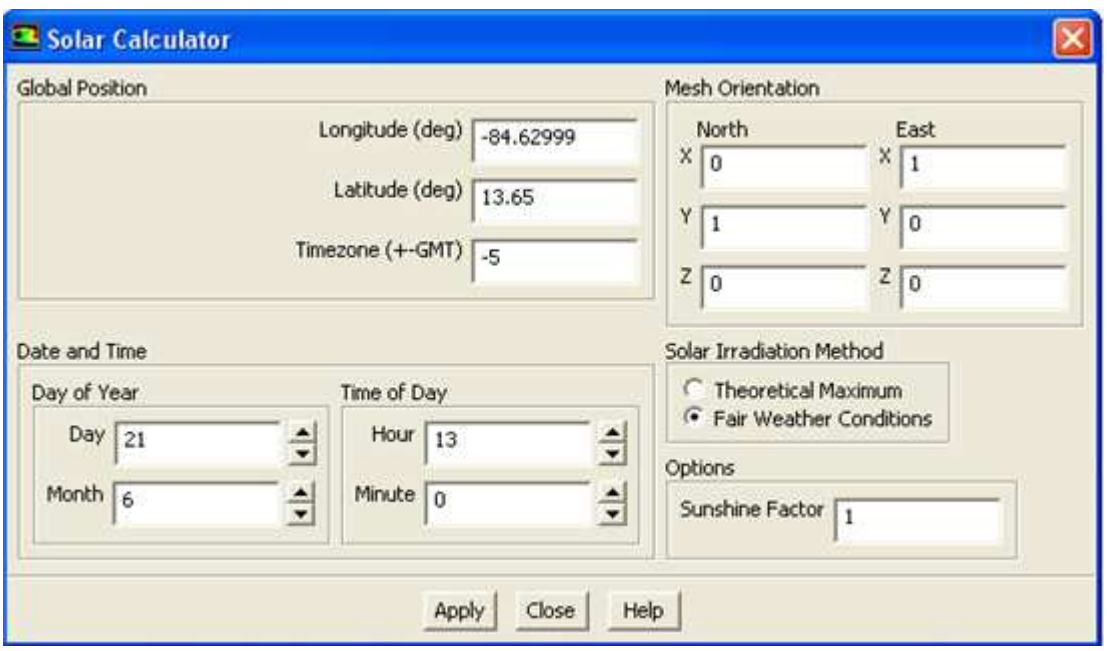
$$\frac{V}{V+IR} \tag{14–19}$$

where V is the visible incident solar radiation, and $V+IR$ is the total incident solar radiation (visible plus infrared).

3. Use the solar calculator to compute solar beam direction and irradiation.

a. Click **Solar Calculator...** in the **Radiation Model** dialog box to open the **Solar Calculator** dialog box (*Figure 14.29* (p. 752)).

Figure 14.29 The Solar Calculator Dialog Box



b. In the **Solar Calculator** dialog box, define the **Global Position** by the following parameters:

i. Enter a real number in degrees for **Longitude**. Values may range from -180 to 180 where negative values indicate the Western hemisphere and positive values indicate the Eastern hemisphere.

ii. Enter a real number for **Latitude** in degrees. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. Enter an integer for **Timezone** that is the local time zone in hours relative to Greenwich Mean Time (+-GMT). This value can range from $+12$ to -12.

Important

Note that you must specify all three **Global Position** parameters for the solar calculator.

c. Define the local **Date and Time** by the following parameters:

i. Enter an integer for **Day** and **Month** under **Day of Year**.

ii. Enter an integer for **Hour** that ranges from 0 to 24 under **Time of Day**. Enter an integer or floating point number for **Minute**.

The time of day is based on a 24-hour clock: 0 hours and 0 minutes corresponds to 12:00 a.m. and 23 hours 59.99 min corresponds to 11:59.99 p.m. For example, if the local time was 12:01:30 a.m., you would enter `0` for **Hour** and `1.5` for **Minute**. If the local time was 4:17 p.m., you would enter `16` for **Hour** and `17` for **Minute**.

d. Define the **Mesh Orientation** as the vectors for North and East in the CFD mesh system of co-ordinates.

e. Select the appropriate **Solar Irradiation Method**. The **Fair Weather Conditions** is the default method.

f. Enter an integer for **Sunshine Fraction** (default = 1).

g. Click **Apply**.

The solar calculator output parameters are computed and the results are reported in the console. The default values are shown below:

```
Fair Weather Conditions:
 Sun Direction Vector: X: -0.0785396, Y: 0.170758, X: 0.982178
 Sunshine Fraction: 1
 Direct Normal Solar Irradiation (at Earth's surface) [W/m^2]:
 881.635
 Diffuse Solar Irradiation - vertical surface: [W/m^2]:
 152.107
 Diffuse Solar Irradiation - horizontal surface: [W/m^2]:
 118.727
 Ground Reflected Solar Irradiation - vertical surface: [W/m^2]:
 96.4649
```

4. For transient simulations, enter the **Time Steps Per Solar Load Update** under **Update Parameters**. The number of time steps that you specify will direct the ANSYS FLUENT solver to update the solar load data for the specified flow-time intervals in the unsteady solution process.

14.3.10.5.3. Setting Boundary Conditions for Solar Loading

Once you have defined the solar parameters for the solar load model (*Setting Up the Solar Load Model* (p. 748)), you will need to set up boundary conditions for boundary zones that will participate in solar loading.

Boundary Conditions

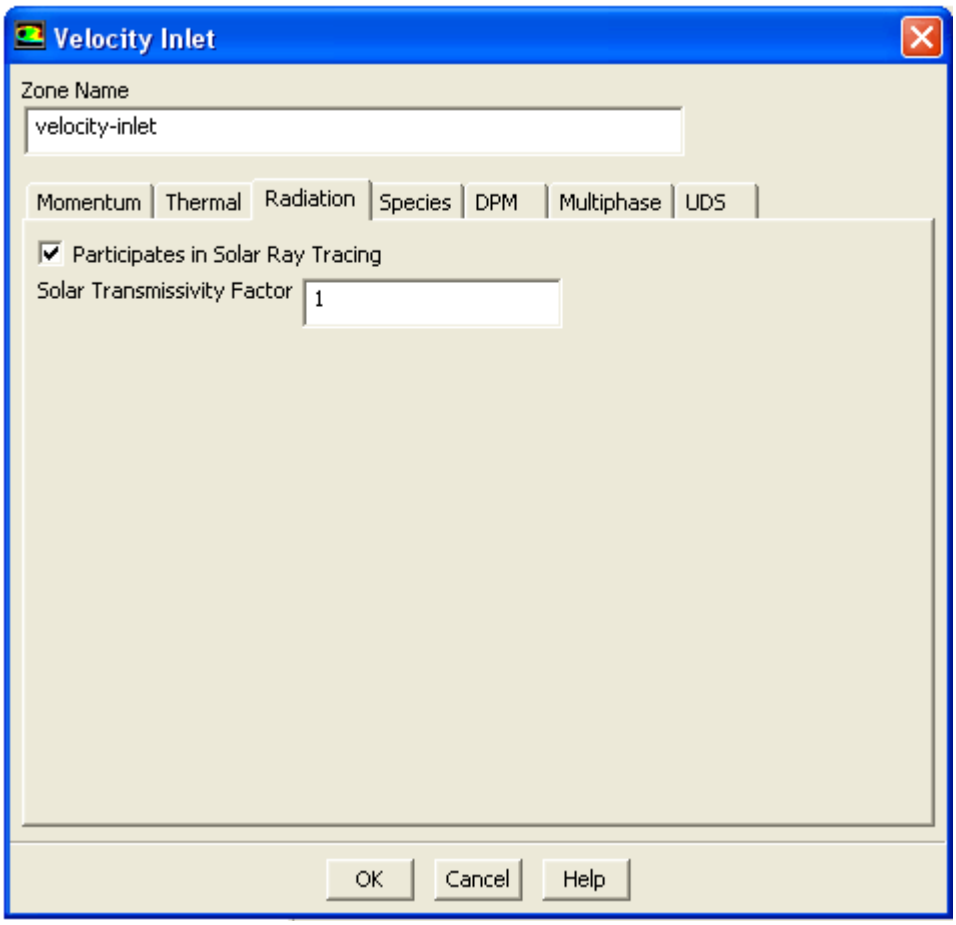
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.10.5.4. Solar Ray Tracing

1. Set the boundary condition for each inlet and exit boundary zone that you want to include in solar loading.

   a. Open the inlet or exit boundary condition dialog box (e.g., **Velocity Inlet**) and click the **Radiation** tab (*Figure 14.30* (p. 754)).

   Figure 14.30 The Velocity Inlet Dialog Box



   b. Enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default). If you deactivate solar ray tracing by disabling this option the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

   c. Enter a value between `0` and `1` for the **Solar Transmissivity Factor**. This will allow you to control the amount of solar irradiation entering the domain. By reducing the solar transmissivity factor from 1 to 0.5, you can effectively cut the total internal energy source entering the domain by half.

   Important

   Note that the solar transmissivity factor is applied to both direct and diffuse solar irradiation components.

   d. Click **OK**.

2. Set the boundary condition for each wall boundary zone that you want to include in solar loading.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Open a **Wall** boundary condition dialog box and click the **Radiation** tab.

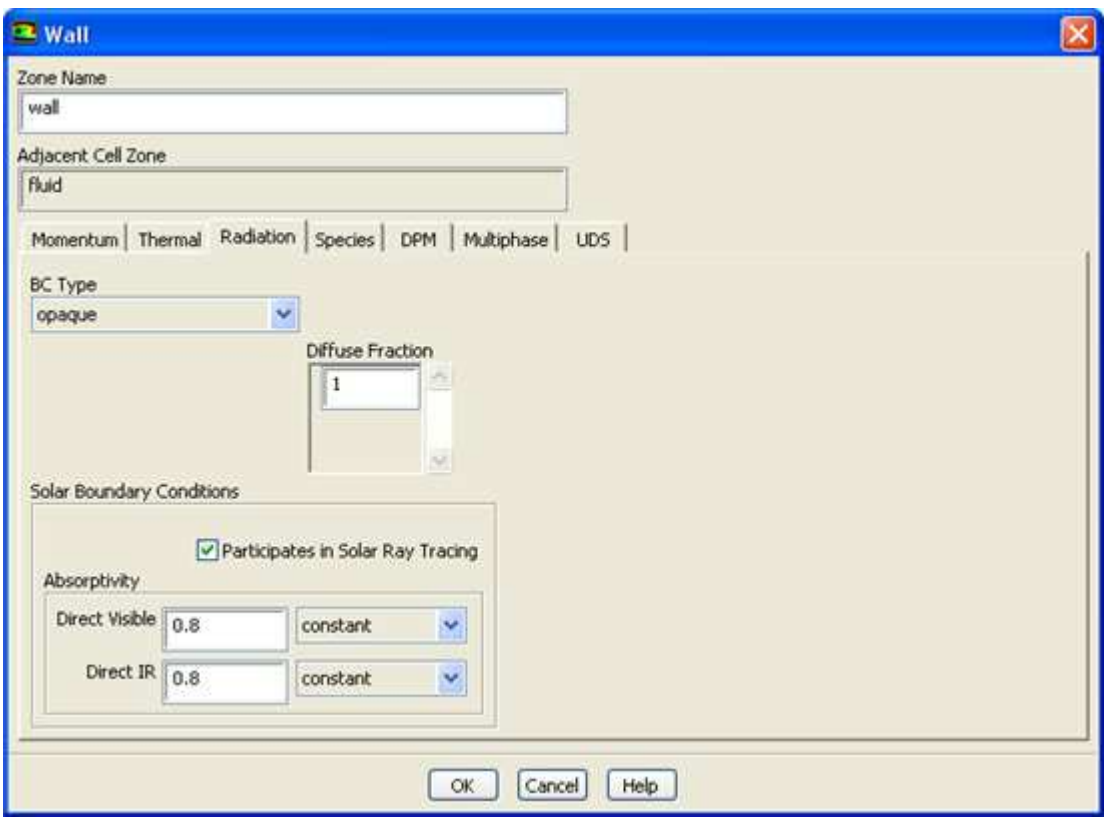
b. Define the wall as **opaque** or **semi-transparent**. An opaque wall will not allow any solar radiation to pass through it, while a semi-transparent surface will allow a portion of the solar radiation to pass through it.)

   i. For an opaque wall, select **opaque** from the drop-down list for **BC Type** (*Figure 14.31* (p. 755)). Then enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default) in the **Solar Boundary Conditions** group box and enter constant values for **Direct Visible** and **Direct IR** absorptivity. Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Important

Absorption in the visible and infrared portions of the spectrum define the surface material for the opaque wall.

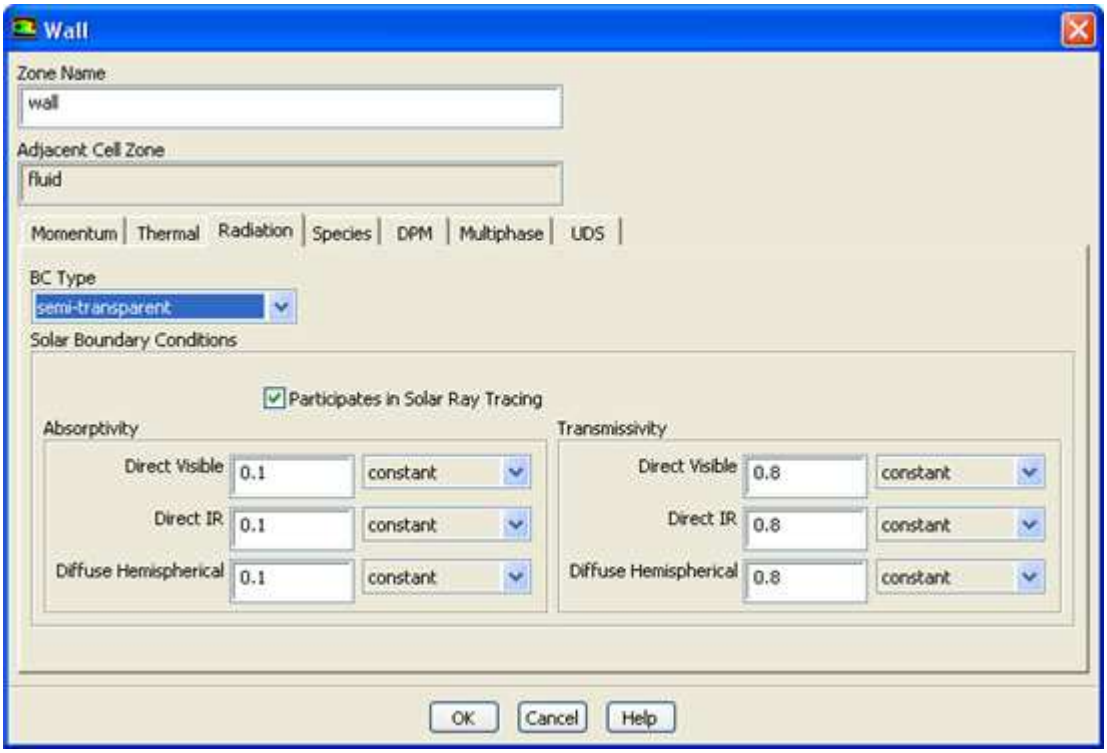
Figure 14.31 The Wall Dialog Box



   ii. For a semi-transparent wall, select **semi-transparent** from the drop-down list for **BC Type** (*Figure 14.32* (p. 756)). Then, enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default) in the **Solar Boundary Conditions** group box and enter constant values for **Direct Visible**, **Direct IR**, and **Diffuse Hemispherical** absorptivity and transmissivity. Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.32 The Wall Dialog Box



Absorption and transmittance in the visible and infrared portions of the spectrum, as well as the "shading" formulation (**Diffuse Hemispherical**), define the surface material for a semi-transparent wall. These parameters are properties of the glazed unit and should be provided by the glazing manufacturer. The direct components are based on normal incident radiation (ANSYS FLUENT adjusts this for the actual angle of incidence). Most manufacturers present this information in a slightly different way so it may be necessary to seek guidance from the supplier. Another useful source of data can be found in the ASHRAE Fundamentals Handbook, chapter on Fenestration.

iii. Click **OK**.

Important

ANSYS FLUENT will calculate the reflectivity as the difference between one and the sum of absorptivity and transmissivity:

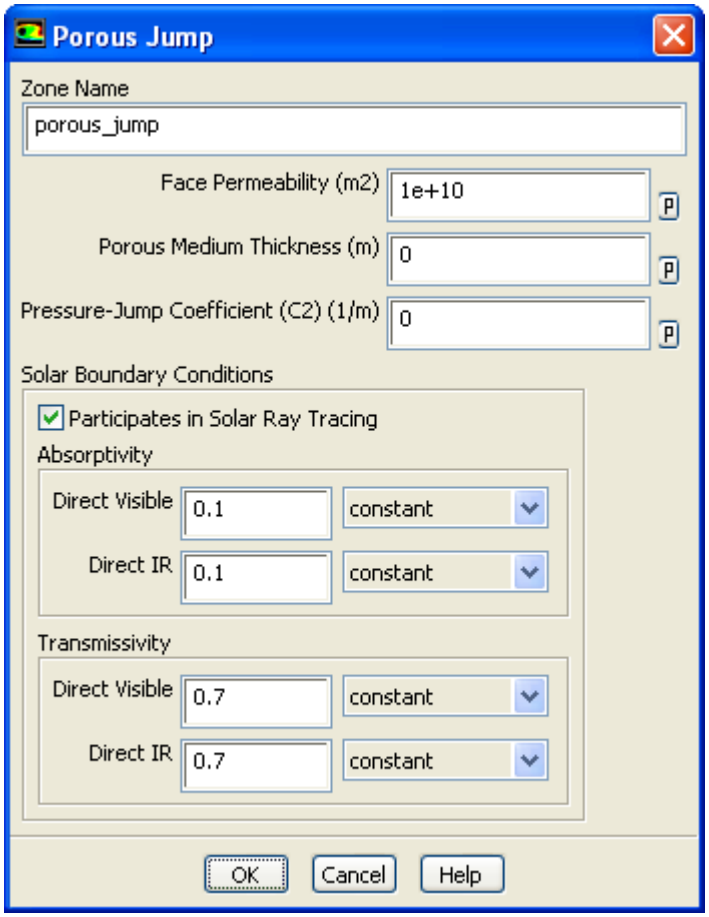
$$reflectivity = 1 - (absorptivity + transmissivity) \quad \textbf{(14–20)}$$

3. Set the boundary condition for each porous jump boundary zone that you want to include in solar loading. You can define the boundary such that it acts as a semi-transparent surface that will allow a portion of the solar radiation to pass through it, along with fluid flow. In this way you can represent a partial opening (e.g., a grate, a grill, or a louver) that is a combination of gaps and (typically) opaque surfaces.

   a. Open the **Porous Jump** boundary condition dialog box (*Figure 14.33* (p. 757)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.33 The Porous Jump Dialog Box



b. Define the **Face Permeability**, **Porous Medium Thickness**, and **Pressure-Jump Coefficient**, as described in *User Inputs for the Porous Jump Model* (p. 338).

c. Define the settings in the **Solar Boundary Conditions** group box.

   i. Enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default). Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

   ii. Enter constant values (between 0 and 1) for **Direct Visible** and **Direct IR** in the **Absorptivity** group box. These values act as multipliers for the visible and infrared portions of the direct solar radiation spectrum, respectively, to account for the absorption of the porous jump.

Important

One reasonable way to estimate the **Direct Visible** and **Direct IR** absorptivity values for a grill, etc., is to use the product of the obstructed area fraction and the surface absorptivity. For example, if 40% of the grill facial area is obstructed with grill slats, and the slats have a surface absorptivity of 0.7, you could estimate the absorptivity as being 0.28.

   iii. Enter constant values (between 0 and 1) for **Direct Visible** and **Direct IR** in the **Transmissivity** group box. These values act as multipliers for the visible and infrared portions of the direct solar radiation spectrum, respectively, to account for the transmissivity of the porous jump.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

One reasonable way to estimate the **Direct Visible** and **Direct IR** transmissivity values for a grill, etc., is to use the open area fraction. For example, if a grill has openings between the slats that amount to 60% of the area, you could estimate the transmissivity as being 0.6.

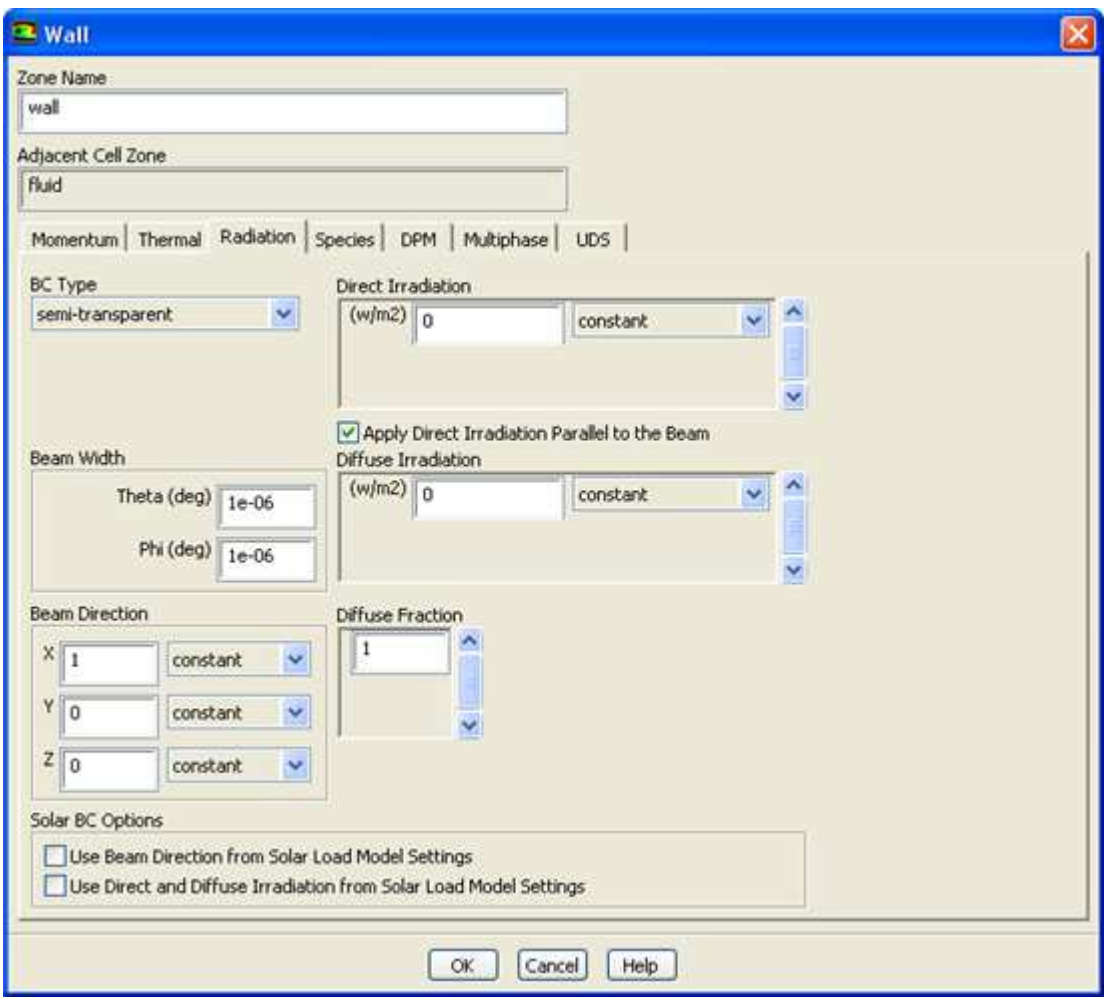
d. Click **OK**.

14.3.10.5.5. DO Irradiation

1. For DO irradiation, all boundary conditions are set up as normal for the DO model, except that now you can select semi-transparent boundary surfaces which will provide a source of solar irradiation.

   a. Open a **Wall** boundary condition dialog box and click the **Radiation** tab (*Figure 14.34* (p. 758)).

Figure 14.34 The Wall Dialog Box



   b. Select **semi-transparent** from the drop-down list for **BC Type**.

   c. Enable the **Use Beam Direction from Solar Load Model Settings** option, under **Solar BC Options**, to have the values for beam direction applied from the **Solar Load Model** settings in the **Radiation** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the sign of the beam direction that is needed for the DO model is opposite the sun direction vector that is entered or derived from the solar parameters. The beam direction in the DO model is the direction of external radiation (e.g., radiation coming from the sun), while the sun direction vector in the solar load model points to the sun. Incident radiation and sun angle always have an opposite sign since they are quantities that are defined from opposite perspectives.

d. Enable the **Use Direct and Diffuse Irradiation from Solar Load Model Settings** option to have the solar calculator output be applied for direct and diffuse irradiation.

   When **Use Direct and Diffuse Irradiation from Solar Load Model Settings** is enabled, the beam width will automatically be set to 0.53 degrees - the angle subtended by the sun.

e. Click **OK**.

14.3.10.5.6. Text Interface-Only Commands

ANSYS FLUENT has provided some additional commands for solar load setup that are only available in the text interface. These commands are present in teh following sections.

14.3.10.5.6.1. Automatically Saving Solar Ray Tracing Data

It is possible to direct ANSYS FLUENT to automatically save solar load data to a generic file that you can examine or use in an external program. This is done by executing the text command `autosave-solar-data` from the text interface.

`define` → `models` → `radiation` → `solar-parameters` → `autosave-solar-data`

1. Enter the **Solar Data File Frequency** (default= 0).
2. Enter the filename, in quotations.
3. Choose to write file in binary format.

The text interface command for `autosave-solar-data` for a file named `solar` and a frequency of 1 is shown below:

```
/define/models/radiation/solar-parameters> autosave-solar-data
 Autosave Solar Data File Frequency [0] 1
 Enter Filename [""] "solar"
```

14.3.10.5.6.2. Automatically Reading Solar Data

When you are executing a transient simulation in parallel ANSYS FLUENT and you want to take solar loading conditions into consideration, you can use the `autoread-solar-data` text command to automatically read the solar load data file you generated during a serial run into parallel ANSYS FLUENT. This is done by executing the text command `autoread-solar-data` from the text interface.

`define` → `models` → `radiation` → `solar-parameters` → `autoread-solar-data`

1. Enter the **Solar Data File Frequency** (default= 0).
2. Enter the filename, in quotations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The text interface command for `autoread-solar-data` for a file named `solar` and a frequency of 1 is shown below:

```
/define/models/radiation/solar-parameters> autoread-solar-data
 Autoread Solar Data File Frequency [0] 1
 Enter Filename [""] "solar"
 Use Binary Format for Reading Data Files [yes]
```

14.3.10.5.6.3. Aligning the Camera Direction With the Position of the Sun

When the solar load model is enabled, you can direct ANSYS FLUENT to align the camera direction with the sun position using the text interface command:

`define` → `models` → `radiation` → `solar-parameters` → `sol-camera-pos`

This command is useful when you are executing a transient simulation and you want to capture an image of your model with solar load parameters displayed (such as solar heat flux) as the sun position changes with time in order to create an animation. See *Postprocessing Solar Load Quantities* (p. 761) for details.

14.3.10.5.6.4. Specifying the Scattering Fraction

You can modify the default scattering fraction (1) using the text interface command:

`define` → `models` → `radiation` → `solar-parameters` → `scattering-fraction`

The scattering fraction is the amount of direct radiation that has been reflected from opaque surfaces (after entering through the transparent surfaces) that will be considered to remain within the space and be evenly distributed among all surfaces. The value is between 0 and 1.

The text interface command for specifying a `scattering-fraction` of 0.5 is shown below:

```
/define/models/radiation/solar-parameters> scattering-fraction
 Scattering Fraction [1] .5
```

14.3.10.5.6.5. Applying the Solar Load on Adjacent Fluid Cells

You can direct ANSYS FLUENT to apply the solar load that is computed from the solar ray tracing algorithm to adjacent fluid cells by issuing the following command at the text interface:

`define` → `models` → `radiation` → `solar-parameters` → `sol-adjacent-fluidcells`

The text interface command is shown below:

```
/define/models/radiation/solar-parameters> sol-adjacent-fluidcells
 Apply Solar Load on adjacent Fluid Cells? [no] y
```

This command allows you to apply solar loads to adjacent fluid cells only, even if solid or shell conduction zones are present. By applying the solar load on adjacent fluid cells, you are overruling the default order of the adjacent cell assignment in ANSYS FLUENT which is shell, solid, fluid.

14.3.10.5.6.6. Specifying Quad Tree Refinement Factor

You can modify the default value (7) for the maximum quad tree refinement factor in the solar ray tracing algorithm using the text command:

`define` → `models` → `radiation` → `solar-parameters` → `quad-tree-parameters`

The text interface command is shown below, when a new maximum refinement value of 10 is specified:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/define/models/radiation/solar-parameters> quad-tree-parameters
 Maximum Quad-Tree Refinement [7] 10
```

14.3.10.5.6.7. Specifying Ground Reflectivity

You can modify the default value (0.2) for the ground reflectivity using the text command:

`define` → `models` → `radiation` → `solar-parameters` → `ground-reflectivity`

Ground reflectivity ρ_g (*Equation 14–18* (p. 747)) includes the contribution of reflected solar radiation from ground surfaces. It is treated as part of the total diffuse solar irradiation when the solar calculator is used in conjunction with the `Diffuse Solar Irradiation` illumination parameter. The default value is 0.2.

```
/define/models/radiation/solar-parameters> ground-reflectivity
 Ground Reflectivity [0.2] 0.5
```

14.3.10.5.6.8. Additional Text Interface Commands

Some solar load commands that are available in the graphical user interface are also made available in the text interface. For example, you can turn the solar load model on using the text command:

`define` → `models` → `radiation` → `solar?`

You can also enter the solar calculator parameters in the text interface by executing the command:

`define` → `models` → `radiation` → `solar-calculator`

Once invoked, you will be prompted to enter the solar calculator input parameters.

To set the illumination parameters, select this option from the `solar-parameters` menu:

`define` → `models` → `radiation` → `solar-parameters` → `illumination-parameters`

And finally, you can direct ANSYS FLUENT to compute the solar load on demand, by issuing the text command:

`define` → `models` → `radiation` → `solar-parameters` → `sol-on-demand`

When the command is initiated, the solar data are written to the console.

14.3.10.6. Postprocessing Solar Load Quantities

The following solar load quantities can be used to visualize the illuminated areas and shadows created by solar radiation.

- solar heat flux (i.e., sum of visible and IR absorbed solar flux on opaque walls)
- absorbed visible and IR solar flux (semi-transparent walls and porous jump boundaries only)
- reflected visible and IR solar flux (semi-transparent walls and porous jump boundaries only)
- transmitted visible and IR solar flux (semi-transparent walls and porous jump boundaries only)

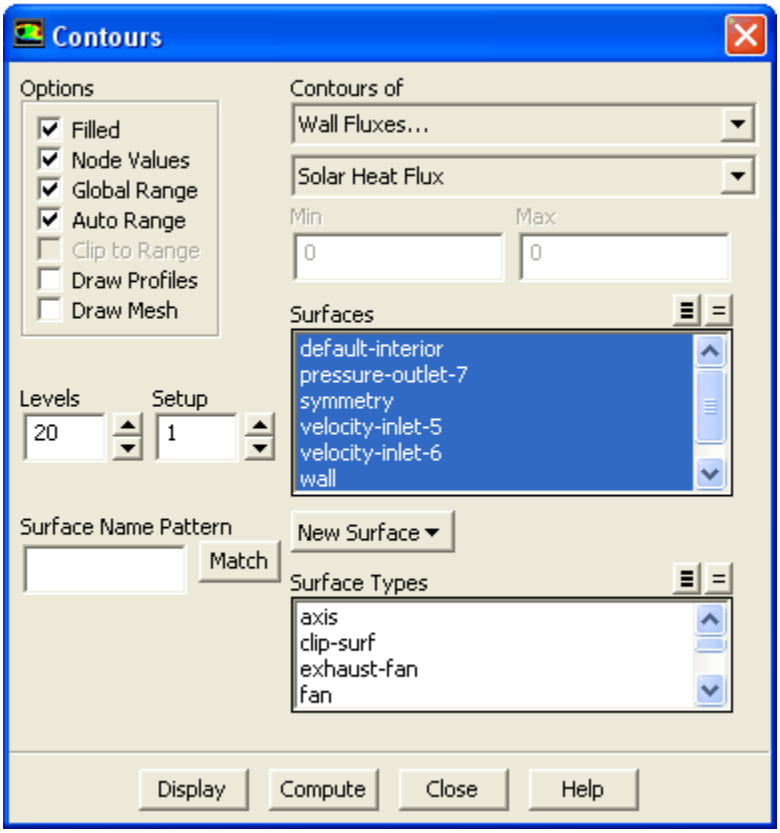
These quantities are available for postprocessing of solar loading at wall boundaries and can be displayed as contours of **Wall Fluxes** in the **Contours** dialog box. For steady-state simulations, the solar flux data is computed at solution initialization and is available for postprocessing. You can also compute the solar load at any time during your ANSYS FLUENT session, after you have set up the model and applied boundary conditions. To compute the solar load on demand, you can issue the `sol-on-demand` command in the text interface (see *Additional Text Interface Commands* (p. 761) for details).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solar heat flux, for example, can be displayed for surfaces using the **Contours** dialog box. A sample dialog is shown below (*Figure 14.35* (p. 762)).

Graphics and Animations → **Contours** → **Set Up...**

Figure 14.35 The Contours Dialog Box



14.3.10.6.1. Solar Load Animation at Different Sun Positions

The solar camera alignment command is useful when you want to take timed pictures of solar loading effects of your model during transient simulations, and later create animations of the image files using an external program. Follow the procedure below.

1. Read (or set up) your transient case file in ANSYS FLUENT.
2. Set up the automatic execution of solution commands in the **Execute Commands** dialog box that will: 1) display solar load parameter graphics, 2) re-position the solar camera such that the view is aligned with the instantaneous sun direction, and 3) generate a picture image file (`.tiff`) during the solution process in the **Execute Commands** dialog box.

   Calculation Activities (Execute Commands) → **Create/Edit...**
3. Initialize and run the solution.
4. Animate the `.tiff` files using an external animation tool.

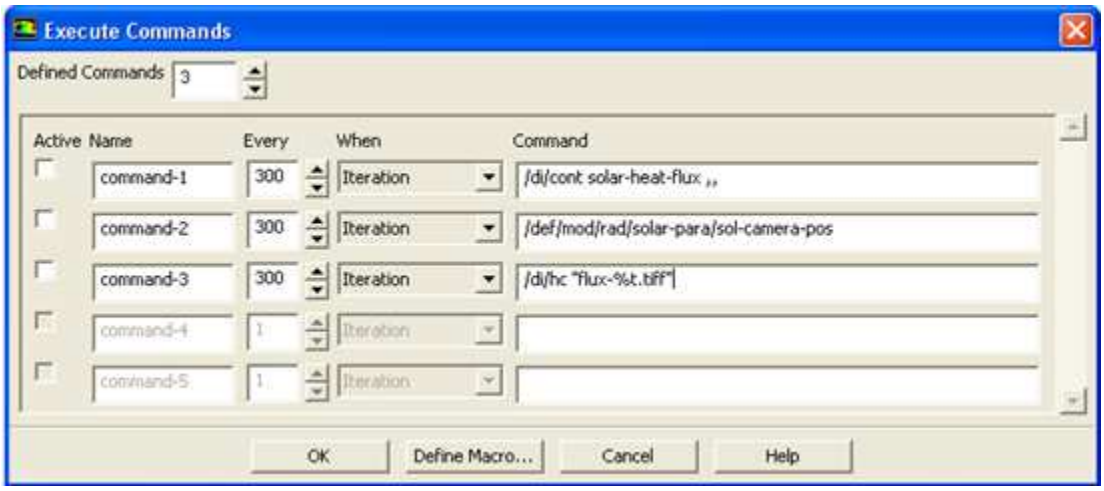
The following commands entered in the **Execute Commands** dialog box will direct ANSYS FLUENT to display contours of solar heat flux, align the camera with the current direction of the sun, and then generate a picture

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

image file (`.tiff`) of the solar heat flux contour every 300 time steps during the unsteady simulation. See *Figure 14.36* (p. 763).

```
/di/cont solar-heat-flux ,,
/def/mod/rad/solar-para/sol-camera-pos
/di/hc "flux-%t.tiff"
```

Figure 14.36 The Execute Commands Dialog Box



14.3.10.6.2. Reporting and Displaying Solar Load Quantities

ANSYS FLUENT provides some additional solar load variables that you can use for postprocessing when your model includes solar ray tracing. You can generate graphical plots or alphanumeric reports of the following variables:

In the **Wall Fluxes...** category:

- **Solar Heat Flux**
- **Transmitted Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Transmitted IR Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Reflected Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Reflected IR Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Absorbed Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Absorbed IR Solar Flux** (semi-transparent walls and porous jump boundaries)

See *Field Function Definitions* (p. 1529) for their definitions.

14.4. Modeling Periodic Heat Transfer

ANSYS FLUENT is able to predict heat transfer in periodically repeating geometries, such as compact heat exchangers, by including only a single periodic module for analysis.

This section discusses streamwise-periodic heat transfer. The treatment of streamwise-periodic flows is discussed in *Periodic Flows* (p. 493), and a description of no-pressure-drop periodic flow is provided in *Periodic Boundary Conditions* (p. 321).

Information about streamwise-periodic heat transfer is presented in the following sections:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.4.1. Overview and Limitations
14.4.2. Theory
14.4.3. Using Periodic Heat Transfer
14.4.4. Solution Strategies for Periodic Heat Transfer
14.4.5. Monitoring Convergence
14.4.6. Postprocessing for Periodic Heat Transfer

14.4.1. Overview and Limitations

14.4.1.1. Overview

As discussed in *Overview and Limitations* (p. 493), streamwise-periodic flow conditions exist when the flow pattern repeats over some length L, with a constant pressure drop across each repeating module along the streamwise direction.

Periodic thermal conditions may be established when the thermal boundary conditions are of the constant wall temperature or wall heat flux type. In such problems, the temperature field (when scaled in an appropriate manner) is periodically fully-developed [*62*] (p. 2212). As for periodic flows, such problems can be analyzed by restricting the numerical model to a single module or periodic length.

14.4.1.2. Constraints for Periodic Heat Transfer Predictions

In addition to the constraints for streamwise-periodic flow discussed in *Limitations for Modeling Streamwise-Periodic Flow* (p. 494), the following constraints must be met when periodic heat transfer is to be considered:

- The pressure-based solver must be used.
- The thermal boundary conditions must be of the specified heat flux or constant wall temperature type. Furthermore, in a given problem, these thermal boundary types cannot be combined: all boundaries must be either constant temperature or specified heat flux. You can, however, include constant-temperature walls and zero-heat-flux walls in the same problem. For the constant-temperature case, all walls must be at the same temperature (profiles are not allowed) or zero heat flux. For the heat flux case, profiles and/or different values of heat flux may be specified at different walls.
- When constant-temperature wall boundaries are used, you cannot include viscous heating effects or any volumetric heat sources.
- In cases that involve solid regions, the regions cannot straddle the periodic plane.
- The thermodynamic and transport properties of the fluid (heat capacity, thermal conductivity, viscosity, and density) cannot be functions of temperature. (You cannot, therefore, model reacting flows.) Transport properties may, however, vary spatially in a periodic manner, and this allows you to model periodic turbulent flows in which the effective turbulent transport properties (effective conductivity, effective viscosity) vary with the (periodic) turbulence field.

Theory (p. 764) and *Using Periodic Heat Transfer* (p. 765) provide more detailed descriptions of the input requirements for periodic heat transfer.

14.4.2. Theory

Streamwise-periodic flow with heat transfer from constant-temperature walls is one of two classes of periodic heat transfer that can be modeled by ANSYS FLUENT. A periodic fully-developed temperature field can also be obtained when heat flux conditions are specified. In such cases, the temperature change between periodic boundaries becomes constant and can be related to the net heat addition from the boundaries as described in this section.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Periodic heat transfer can be modeled only if you are using the pressure-based solver.

14.4.2.1. Definition of the Periodic Temperature for Constant- Temperature Wall Conditions

For the case of constant wall temperature, as the fluid flows through the periodic domain, its temperature approaches that of the wall boundaries. However, the temperature can be scaled in such a way that it behaves in a periodic manner. A suitable scaling of the temperature for periodic flows with constant-temperature walls is [62] (p. 2212)

$$\theta = \frac{T(\vec{r}) - T_{wall}}{T_{bulk,inlet} - T_{wall}} \tag{14–21}$$

The bulk temperature, $T_{bulk,inlet}$, is defined by

$$T_{bulk,inlet} = \frac{\int_A T \left| \rho \vec{v} \cdot d\vec{A} \right|}{\int_A \left| \rho \vec{v} \cdot d\vec{A} \right|} \tag{14–22}$$

where the integral is taken over the inlet periodic boundary (A). It is the scaled temperature, θ, which obeys a periodic condition across the domain of length L.

14.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions

When periodic heat transfer with heat flux conditions is considered, the form of the unscaled temperature field becomes analogous to that of the pressure field in a periodic flow:

$$\frac{T\left(\vec{r}+\vec{L}\right) - T(\vec{r})}{L} = \frac{T\left(\vec{r}+2\vec{L}\right) - T\left(\vec{r}+\vec{L}\right)}{L} = \sigma\,. \tag{14–23}$$

where $\vec{L}$ is the periodic length vector of the domain. This temperature gradient, σ, can be written in terms of the total heat addition within the domain, Q, as

$$\sigma = \frac{Q}{\dot{m} c_p L} = \frac{T_{bulk,exit} - T_{bulk,inlet}}{L} \tag{14–24}$$

where $\dot{m}$ is the specified or calculated mass flow rate.

14.4.3. Using Periodic Heat Transfer

A typical calculation involving both streamwise-periodic flow and periodic heat transfer is performed in two parts. First, the periodic velocity field is calculated (to convergence) without consideration of the temperature

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

field. Next, the velocity field is frozen and the resulting temperature field is calculated. These periodic flow calculations are accomplished using the following procedure:

1. Set up a mesh with translationally periodic boundary conditions.
2. Input constant thermodynamic and molecular transport properties.
3. Specify either the periodic pressure gradient or the net mass flow rate through the periodic boundaries.
4. Compute the periodic flow field, solving momentum, continuity, and (optionally) turbulence equations.
5. Specify the thermal boundary conditions at walls as either heat flux or constant temperature.
6. Define an inlet bulk temperature.
7. Solve the energy equation (only) to predict the periodic temperature field.

In order to model the periodic heat transfer, you will need to set up your periodic model in the manner described in *User Inputs for the Pressure-Based Solver* (p. 494) for periodic flow models with the pressure-based solver, noting the restrictions discussed in *Limitations for Modeling Streamwise-Periodic Flow* (p. 494) and *Constraints for Periodic Heat Transfer Predictions* (p. 764). In addition, you will need to provide the following inputs related to the heat transfer model:

1. Activate solution of the energy equation in the **Energy** dialog box.

   Models → **Energy** → **Edit...**

2. Define the thermal boundary conditions according to one of the following procedures:

   Boundary Conditions

   - If you are modeling periodic heat transfer with specified-temperature boundary conditions, set the wall temperature T_{wall} for all wall boundaries in their respective **Wall** dialog box. Note that all wall boundaries must be assigned the same temperature and that the entire domain (except the periodic boundaries) must be "enclosed" by this fixed-temperature condition, or by symmetry or adiabatic (q=0) boundaries.
   - If you are modeling periodic heat transfer with specified-heat-flux boundary conditions, set the wall heat flux in the **Wall** dialog box for each wall boundary. You can define different values of heat flux on different wall boundaries, but you should have no other types of thermal boundary conditions active in the domain.

3. Define solid regions, if appropriate, according to one of the following procedures:

   Cell Zone Conditions

   - If you are modeling periodic heat transfer with specified-temperature conditions, conducting solid regions can be used within the domain, provided that on the perimeter of the domain they are enclosed by the fixed-temperature condition. Heat generation within the solid regions is not allowed when you are solving periodic heat transfer with fixed-temperature conditions.
   - If you are modeling periodic heat transfer with specified-heat-flux conditions, you can define conducting solid regions at any location within the domain, including volumetric heat addition within the solid, if desired.

4. Set constant material properties (density, heat capacity, viscosity, thermal conductivity), *not* temperature-dependent properties, using the **Create/Edit Materials** dialog box.

   Materials

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Specify the **Upstream Bulk Temperature** in the **Periodic Conditions** dialog box.

   Boundary Conditions → **Periodic Conditions...**

   Important

   If you are modeling periodic heat transfer with specified-temperature conditions, the bulk temperature should not be equal to the wall temperature, since this will give you the trivial solution of constant temperature everywhere.

6. Set the solution parameters as described in *Solution Strategies for Periodic Heat Transfer* (p. 767).
7. Run the solution and monitor the convergence as described in *Monitoring Convergence* (p. 767).
8. Postprocess the results as described in *Postprocessing for Periodic Heat Transfer* (p. 768).

14.4.4. Solution Strategies for Periodic Heat Transfer

After completing the inputs described in *Using Periodic Heat Transfer* (p. 765), you can solve the flow and heat transfer problem to convergence. The most efficient approach to the solution, however, is a sequential one in which the periodic flow is first solved without heat transfer and then the heat transfer is solved leaving the flow field unaltered. This sequential approach is accomplished as follows:

1. Disable solution of the energy equation in the **Equations** dialog box, accessed via the **Solution Controls** task page.

   Solution Controls → **Equations...**

2. Solve the remaining equations (continuity, momentum, and, optionally, turbulence parameters) to convergence to obtain the periodic flow field.

   Important

   When you initialize the flow field before beginning the calculation, use the mean value between the inlet bulk temperature and the wall temperature for the initialization of the temperature field.

3. Return to the **Solution Controls** task page and turn off solution of the flow equations and turn on the energy solution.
4. Solve the energy equation to convergence to obtain the periodic temperature field of interest.

While you can solve your periodic flow and heat transfer problems by considering both the flow and heat transfer simultaneously, you will find that the procedure outlined above is more efficient.

14.4.5. Monitoring Convergence

If you are modeling periodic heat transfer with specified-temperature conditions, you can monitor the value of the bulk temperature ratio

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\theta = \frac{T_{wall} - T_{bulk,inlet}}{T_{wall} - T_{bulk,exit}} \tag{14–25}$$

during the calculation using the **Statistic Monitors** dialog box to ensure that you reach a converged solution. Select **per/bulk-temp-ratio** as the variable to be monitored. See *Monitoring Statistics* (p. 1277) for details about using this feature.

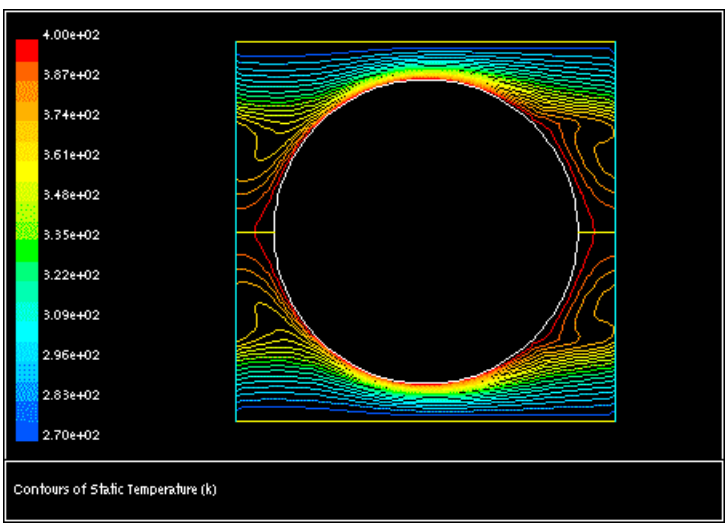
14.4.6. Postprocessing for Periodic Heat Transfer

The actual temperature field predicted by ANSYS FLUENT in periodic models will not be periodic, and viewing the temperature results during postprocessing will display this actual temperature field ($T(\vec{r})$ of *Equation 14–21* (p. 765)). The displayed temperature may exhibit values outside the range defined by the inlet bulk temperature and the wall temperature. This is permissible since the actual temperature profile at the inlet periodic face will have temperatures that are higher or lower than the inlet bulk temperature.

Static Temperature is found in the **Temperature...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

Figure 14.37 (p. 768) shows the temperature field in a periodic heat exchanger geometry.

Figure 14.37 Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 15: Modeling Heat Exchangers

ANSYS FLUENT provides two heat exchanger models: the macro (ungrouped and grouped) models and the dual cell model. The macro model allows you to choose between two heat transfer models, namely the **simple-effectiveness-model** and the number-of-transfer-units (NTU) model. The models can be used to compute auxiliary fluid inlet temperature for a fixed heat rejection or total heat rejection for a fixed auxiliary fluid inlet temperature. For the **simple-effectiveness-model**, the auxiliary fluid may be single-phase or two-phase. The dual cell model uses the NTU method for heat transfer calculations. This model allows the solution of auxiliary flow on a separate mesh (other than the primary fluid mesh), unlike the macro model, where the auxiliary flow is modeled as 1-D flow. The dual cell model also offers more flexibility as far as the shape of the heat exchanger is concerned, and overcomes some of the major limitations present in the macro model.

For theoretical information about the various heat exchanger models, please refer to "Heat Exchangers" in the Theory Guide.

The following sections contain information about the heat exchanger models:

15.1. Overview and Restrictions of the Macro Heat Exchanger Models
15.2. Overview and Restrictions of the Dual Cell Model
15.3. Using the Ungrouped Macro Heat Exchanger Model
15.4. Using the Grouped Macro Heat Exchanger Model
15.5. Using the Dual Cell Heat Exchanger Model
15.6. Postprocessing for the Heat Exchanger Model
15.7. Useful Reporting TUI Commands

15.1. Overview and Restrictions of the Macro Heat Exchanger Models

To use the heat exchanger models, you must define one or more fluid zone(s) to represent the heat exchanger core. Typically, the fluid zone is sized to the dimension of the core itself. As part of the setup procedure, you will define the auxiliary fluid path, the number of macros, and the physical properties and operating conditions of the core (pressure drop parameters, heat exchanger effectiveness, auxiliary fluid flow rate, etc.).

Important

It is highly recommended that the free-form **Tet** mesh is not used in the macro heat exchanger model. Instead, evenly distributed **Hex/Wedge** cells should be used for improved accuracy and a more robust solution process.

Additional information about macro heat exchangers can be found in Overview of the Macro Heat Exchanger Models in the Theory Guide. For the theory behind these models, refer to Macro Heat Exchanger Model Theory in the Theory Guide.

ANSYS FLUENT provides two heat transfer models: the default **ntu-model** and the **simple-effectiveness-model**. The **simple-effectiveness-model** interpolates the effectiveness from the velocity vs effectiveness curve that you provide. For the **ntu-model**, ANSYS FLUENT calculates the effectiveness, ε, from the NTU value that is calculated by ANSYS FLUENT from the heat transfer data provided by the user in tabular format.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT will automatically convert this heat transfer data to a primary fluid mass flow rate vs NTU curve (this curve will be piecewise linear). This curve will be used by ANSYS FLUENT to calculate the NTU for macros based on their size and primary fluid flow rate.

The **ntu-model** provides the following features:

- The model can be used to check heat capacity for both the primary and the auxiliary fluid and takes the lesser of the two for the calculation of heat transfer.
- The model can be used to model heat transfer to the primary fluid from the auxiliary fluid and vice versa.
- The model can be used to model primary fluid-side reverse flow.
- The model can be used with variable density of the primary fluid.
- The model can be used in either the serial or parallel ANSYS FLUENT solvers.
- Transient profiles can be used for the coolant inlet temperature and for total heat rejection.
- Transient profiles can be used for auxiliary mass flow rates.

The **simple-effectiveness-model** provides the following features:

- The model can be used to model heat transfer from the auxiliary fluid to the fluid.
- The auxiliary fluid properties can be a function of pressure and temperature, thus allowing phase change of the auxiliary fluid.
- The model can be used by serial as well as parallel solvers.
- The model can be used to make a network of heat exchangers using a heat exchanger group (*Using the Grouped Macro Heat Exchanger Model* (p. 781)).
- Transient profiles can be used for the coolant inlet temperature and for total heat rejection.
- Transient profiles can be used for auxiliary mass flow rates.

For additional information, please see the following section:
15.1.1. Restrictions

15.1.1. Restrictions

The following restrictions exist for the macro heat exchanger models:

- The core must be approximately rectangular in shape.
- The primary fluid streamwise direction (see Equation 6–1 in the Theory Guide) must be aligned with one of the three orthogonal axes defined by the rectangular core.
- Flow acceleration effects are neglected in calculating the pressure loss coefficient.
- For the **simple-effectiveness-model**, the primary fluid capacity rate must be less than the auxiliary fluid capacity rate.
- Auxiliary fluid phase change cannot be modeled using the **ntu-model**.
- The macro-based method requires that an equal number of cells reside in each macro of equal size and shape.
- Coolant flow is assumed to be 1-D.
- The pass width has to be uniform.
- Accuracy is not guaranteed when the mesh is not structured or layered.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Accuracy is not guaranteed when there is upstream diffusion of temperature at the inlet/outlet of the core.
- Non-conformal meshes cannot be attached to the inlet/outlet of the core. An extra layer has to be created to avoid it.

15.2. Overview and Restrictions of the Dual Cell Model

The macro model is quite suitable for thin rectangular heat exchanger cores, where the pass-to-pass is perpendicular to the primary flow direction and the auxiliary flow is uniform. Moreover, the mesh should be uniform and structured. However, many practical heat exchangers have a non-rectangular core and the auxiliary fluid, before reaching the core, may pass through arbitrary shaped inlet tanks, which make them highly non-uniform. It is quite possible that due to the complex shape of the core and or ease of meshing, the structured mesh may not be the obvious choice. These shortcomings of the macro model can be easily overcome by using the dual cell heat exchanger model. This model allows the solution of both the primary and auxiliary flow on separate co-located meshes and couples the two flows only through heat transfer at the heat exchanger core.

For theoretical information about this model, refer to The Dual Cell Model in the Theory Guide.

15.2.1. Restrictions

The following restrictions exist for the dual cell heat exchanger models:

- Heat transfer calculations are based on the NTU method only.
- In the case of a heat exchanger in which the primary and auxiliary meshes are not identical, heat transfer may be non-conservative (i.e., the heat lost by the hot fluid may not equal the heat gained by the cold fluid). To minimize the difference in heat transfer, the topology and size of the primary and auxiliary cells should be similar.

15.3. Using the Ungrouped Macro Heat Exchanger Model

The heat exchanger model settings may be written into and read from the boundary conditions file (*Reading and Writing Boundary Conditions* (p. 72)) using the text commands, `file/write-settings` and `file/read-settings`, respectively. Otherwise, the steps for setting up the ungrouped macro heat exchanger model is as follows:

1. Enable the calculation of energy in the *Energy Dialog Box* (p. 1645).

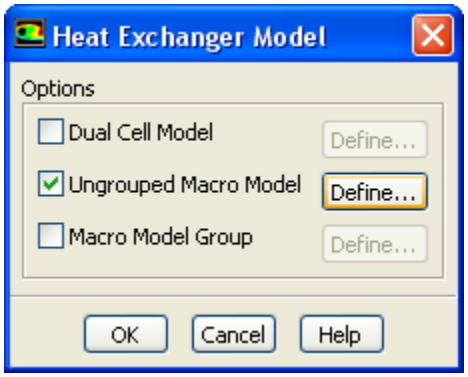
   Models → **Energy** → **Edit...**

2. Enable the **Ungrouped Macro Model** option and click the **Define...** button in the *Heat Exchanger Model Dialog Box* (p. 1666) (*Figure 15.1* (p. 772)) to access the **Ungrouped Macro Heat Exchanger** dialog box.

   Models → **Heat Exchanger** → **Edit...**

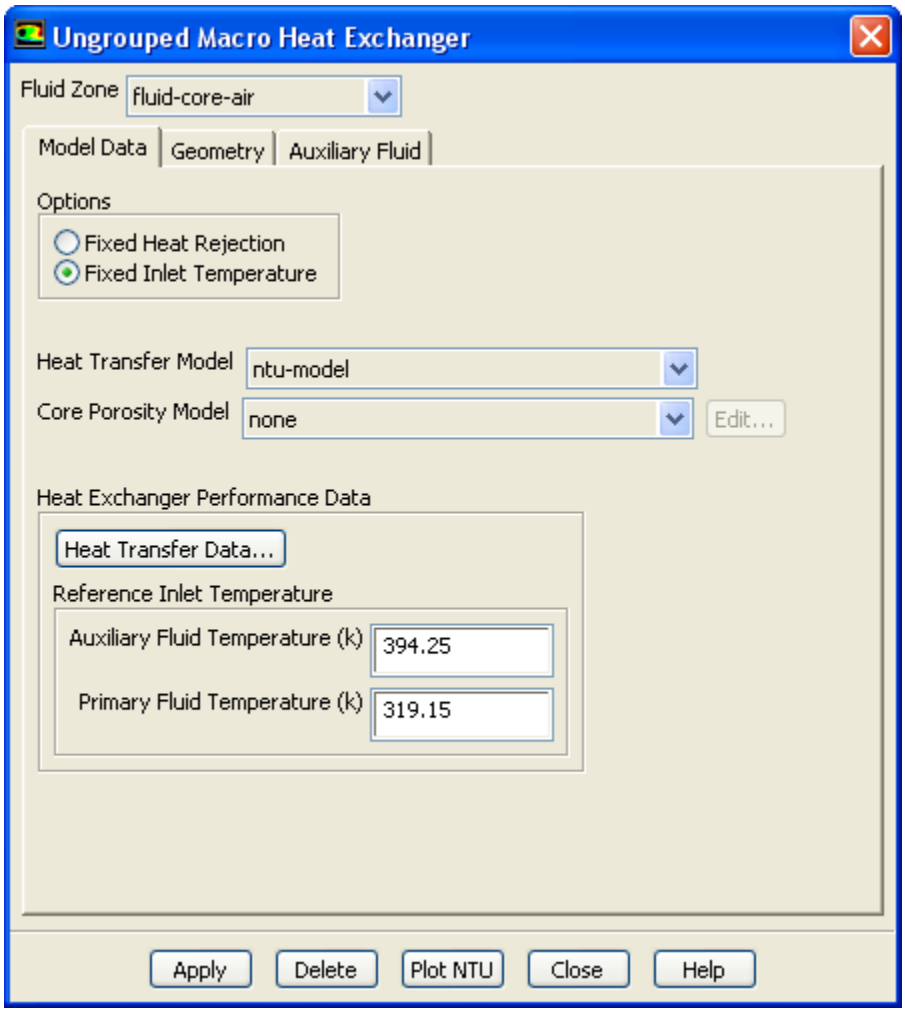
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.1 The Heat Exchanger Model Dialog Box



3. Specify the heat exchanger inputs in the **Ungrouped Macro Heat Exchanger** dialog box.

Figure 15.2 The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab



a. In the **Fluid Zone** drop-down list, select the fluid zone representing the heat exchanger core.

b. Under the **Model Data** tab, choose **Fixed Heat Rejection** or **Fixed Inlet Temperature**, as required (*Figure 15.2* (p. 772)).

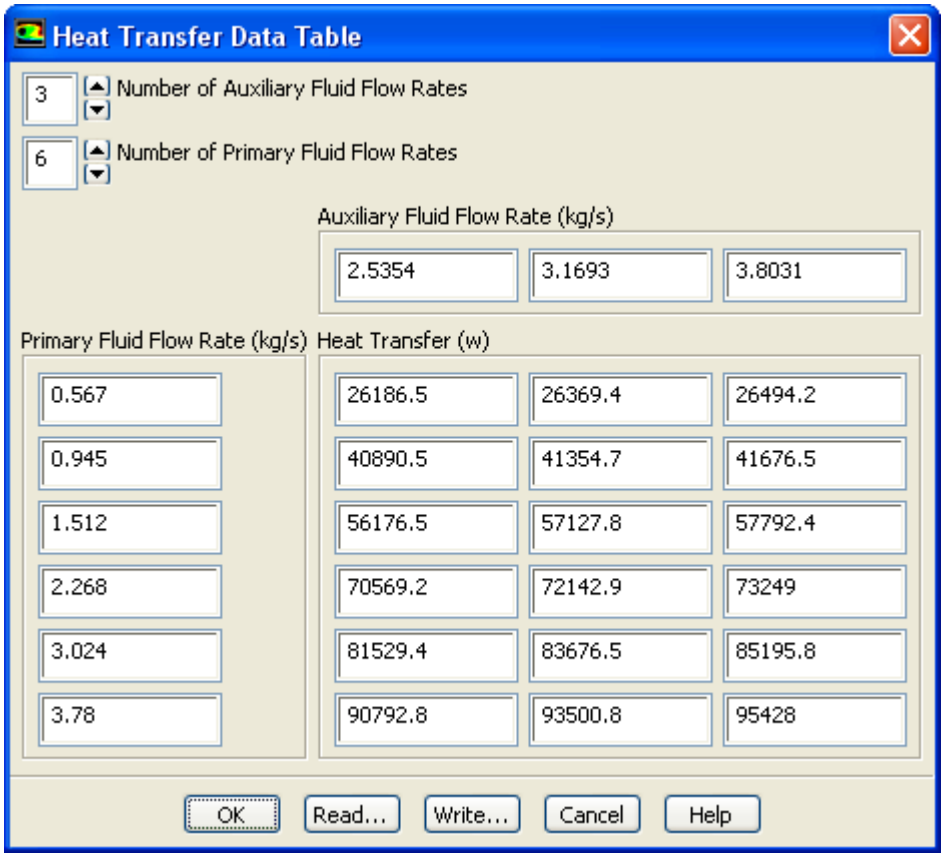
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Specify the heat exchanger model as either the default **ntu-model** or the **simple-effectiveness-model**.

d. Specify the **Core Porosity Model** if you want ANSYS FLUENT to use the pressure loss coefficient function to automatically compute (and update) the porous media coefficients in the cell zones condition dialog box, as described in Streamwise Pressure Drop in the Theory Guide. More information is available in *Setting the Pressure-Drop Parameters and Effectiveness* (p. 779).

e. If the **ntu-model** is chosen, a **Heat Transfer Data...** button will appear under **Heat Exchanger Performance Data**. Clicking the **Heat Transfer Data...** button will open up the *Heat Transfer Data Table Dialog Box* (p. 1670) with information on the fluid flow rates and heat transfer data (*Figure 15.3* (p. 773)). More information is available in *Specifying Heat Exchanger Performance Data* (p. 776).

Figure 15.3 The Heat Transfer Data Table Dialog Box for the NTU Model



f. Enter the **Auxiliary Fluid Temperature** and the Primary Fluid Temperature for the **ntu-model**. These are the fixed inlet temperatures at which the test was performed to obtain the heat transfer data.

g. If the **simple-effectiveness-model** is chosen, then clicking the **Velocity Effectiveness Curve...** button, under the **Heat Exchanger Performance Data**, allows you to set the velocity and corresponding effectiveness for each point. More information is available in *Specifying Heat Exchanger Performance Data* (p. 776).

h. In the **Geometry** tab, define the macro mesh using the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass** fields. The **Number of Rows/Pass** is along the auxiliary flow direction (height) and the **Number of Columns/Pass** is defined in the pass-to-pass (width) direction. Also, enter the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction**. You may want to snap the plane tool to either the inlet or outlet of the heat exchanger using the **Update from Plane Tool**. Note that the plane tool must be attached exactly and oriented so that

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

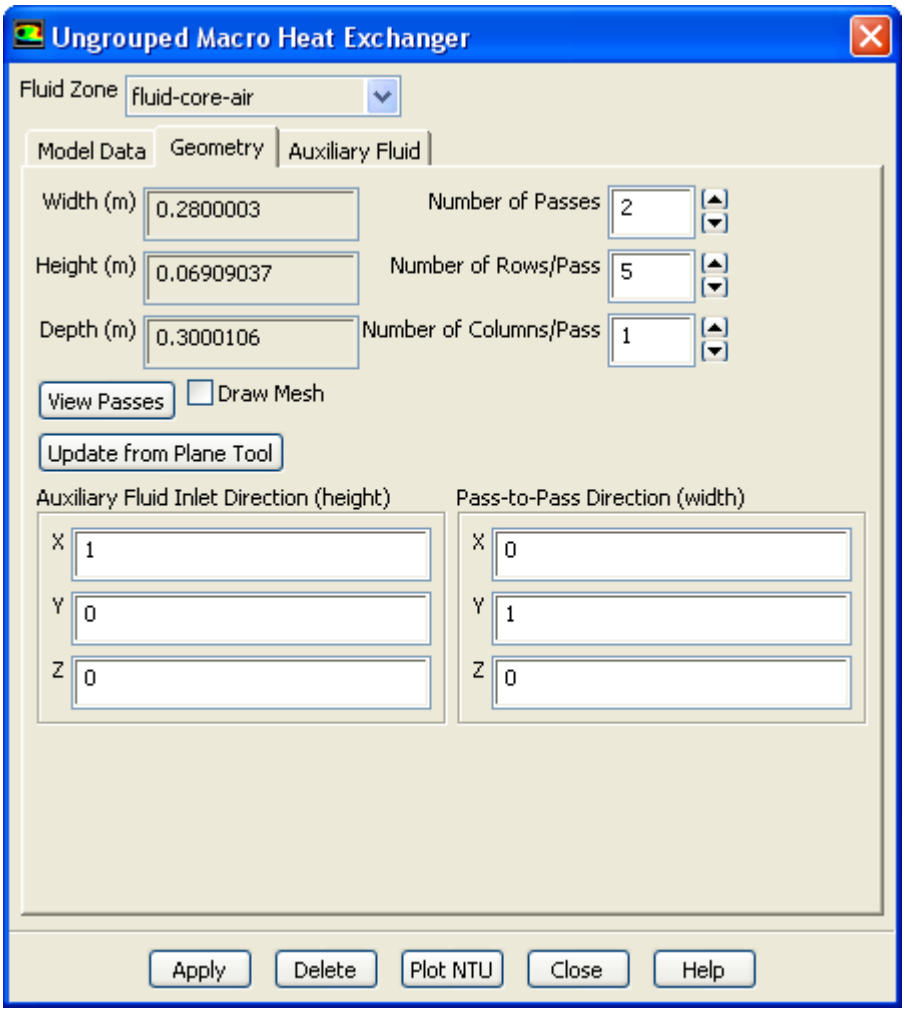
its green arrow points in the auxiliary flow direction, and its blue arrow points in the pass-to-pass direction.

Important

To attach the plane tool exactly, exact coordinates (printed in the console by probing) of the three corner nodes must be entered in the plane tool (x0, x1, x2) in a specific order. Also, note that x0 to x1 is the auxiliary flow direction and x1 to x2 is the pass-to-pass direction.

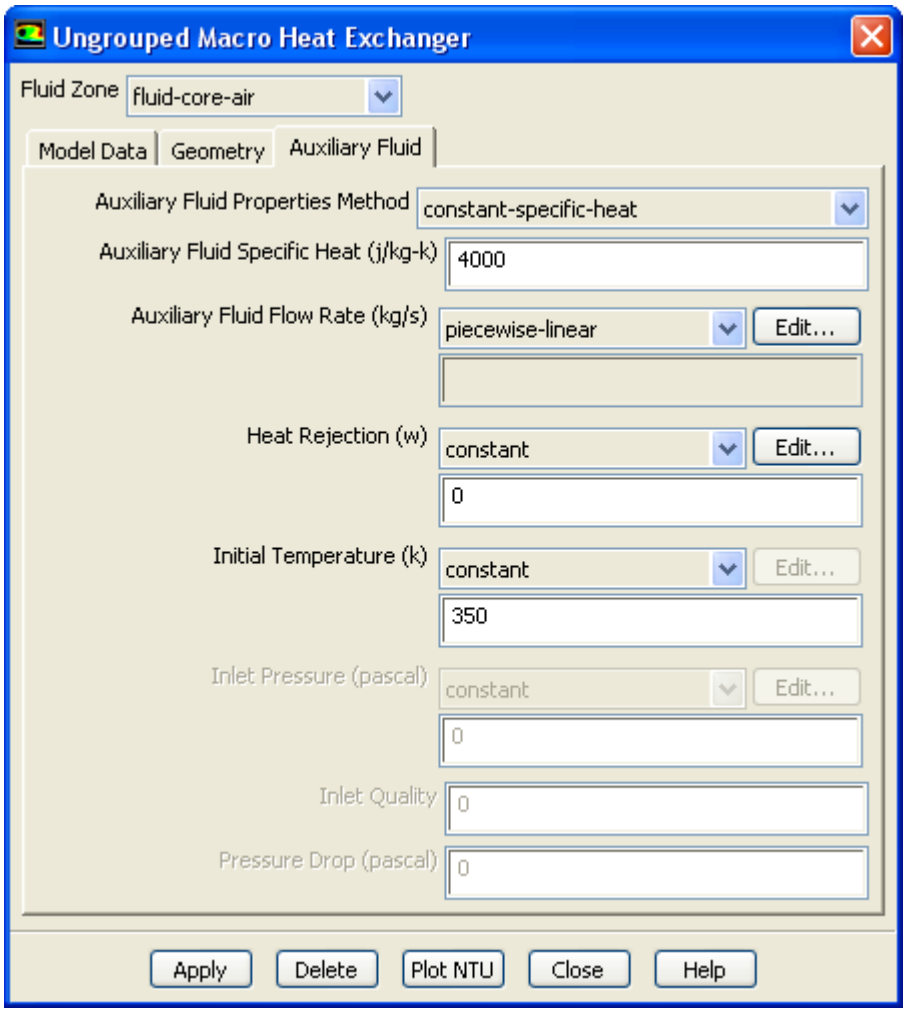
More information is available in *Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions* (p. 777) and *Defining the Macros* (p. 777).

Figure 15.4 The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Geometry Tab



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.5 The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Auxiliary Fluid Tab



i. In the **Auxiliary Fluid** tab, specify the **Auxiliary Fluid Properties Method**, either as a **constant-specific-heat** or as a **user-defined-enthalpy**.

j. **Auxiliary Fluid Flow Rate**, **Heat Rejection**, **Inlet Temperature**, and **Inlet Pressure** can be provided as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time. If **user-defined-enthalpy** is selected as the **Auxiliary Fluid Properties Method**, you will need to specify the **Inlet Quality** and the **Pressure Drop**. More information is available in *Specifying the Auxiliary Fluid Properties and Conditions* (p. 778).

k. Click **Apply** in the **Ungrouped Macro Heat Exchanger** dialog box to save all the settings. Once you click the **Apply** button, the NTU matrix will be computed from the raw data. Therefore, make sure you click **Apply** at the very end of your setup.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

When you click **Apply**, look for any error or warning message in the ANSYS FLUENT console. Some of the common errors you may see displayed are due to the NTU computations not converging. In such cases, check that

- you have entered the data correctly
- the values of the data are reasonable
- the operating condition for the auxiliary fluid flow rate is not too far from the range of the heat transfer data

Other error messages you may encounter may be due to macros not getting any cells assigned to it. In such cases, make sure that

- the heat exchanger core is rectangular
- the directions are correct
- you are using uniformly spaced cells in both directions
- the mesh is either a hexahedra or wedge and is structured

l. Repeat steps (a)–(k) for any other heat exchanger fluid zones.

To use multiple fluid zones to define a single heat exchanger, or to connect the auxiliary fluid flow path among multiple heat exchangers, see *Using the Grouped Macro Heat Exchanger Model* (p. 781).

For additional information, please see the following sections:

15.3.1. Selecting the Zone for the Heat Exchanger
15.3.2. Specifying Heat Exchanger Performance Data
15.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions
15.3.4. Defining the Macros
15.3.5. Specifying the Auxiliary Fluid Properties and Conditions
15.3.6. Setting the Pressure-Drop Parameters and Effectiveness

15.3.1. Selecting the Zone for the Heat Exchanger

Choose the fluid zone for which you want to define a heat exchanger in the **Fluid Zone** drop-down list.

15.3.2. Specifying Heat Exchanger Performance Data

Based on the heat transfer model you choose in the **Model Data** tab, some performance data must be entered for the heat exchanger.

- **ntu-model**: For the **ntu-model** you will provide the heat transfer for different primary and auxiliary fluid flow rates. Click the **Heat Transfer Data...** button to open up a tabular dialog box. Set the Number of auxiliary flow rates and primary fluid flow rates. The dialog box will resize itself accordingly. You will need to provide various primary fluid flow rates and auxiliary fluid flow rates and the corresponding heat transfer values. You may write this data to a file that can be read later.
- **simple-effectiveness-model**: For this model, you will need to provide velocity versus effectiveness data. To provide this you can click the **Velocity Effectiveness Curve...** button. This will open up a tabular dialog box. In this dialog box, you can set the number of points in the curve, then you can provide velocities and corresponding effectiveness values. This data can be written to a file and read back.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions

To define the auxiliary fluid direction and flow path, you will specify direction vectors for the **Auxiliary Fluid Inlet Direction** and the **Pass-to-Pass Direction** in the **Geometry** tab. *Figure 15.6* (p. 777) shows these directions relative to the macros.

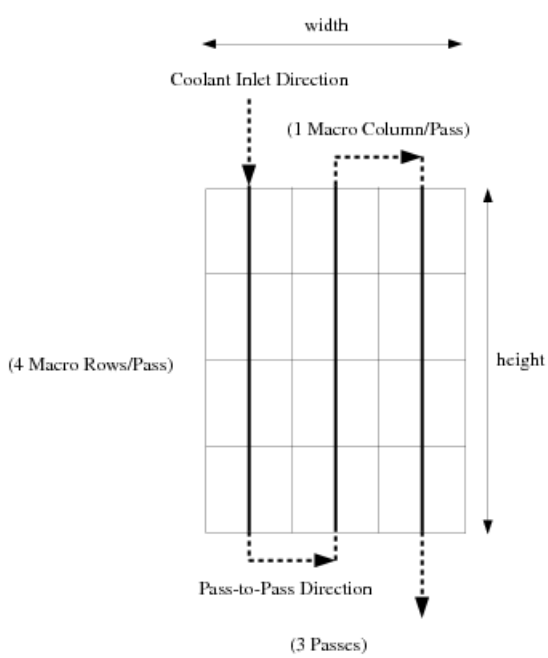
For some problems in which the principal axes of the heat exchanger core are not aligned with the coordinate axes of the domain, you may not know the auxiliary fluid inlet and pass-to-pass direction vectors a priori. In such cases, you can use the plane tool as follows to help you to determine these direction vectors.

1. "Snap" the plane tool onto the boundary of the heat exchanger core. (Follow the instructions in *Initializing the Plane Tool* (p. 1367) for initializing the tool to a position on an existing surface.)
2. Translate and rotate the axes of the tool appropriately until they are aligned with the principal directions of the heat exchanger core. The depth direction is determined by the red axis, the height direction by the green axis, and the width direction by the blue axis.
3. Once the axes are aligned, click the **Update from Plane Tool** button in the **Ungrouped Macro Heat Exchanger** dialog box. The directional vectors will be set automatically. (Note that the **Update from Plane Tool** button will also set the height, width, and depth of the heat exchanger core.)

15.3.4. Defining the Macros

As discussed in *Overview and Restrictions of the Macro Heat Exchanger Models* (p. 769), the fluid zone representing the heat exchanger core is split into macros. Macros are constructed based on the specified number of passes, the number of macro rows per pass, the number of macro columns per pass, and the corresponding auxiliary fluid inlet and pass-to-pass directions (see *Figure 15.6* (p. 777)). Macros are numbered from 0 to $(n-1)$ in the direction of auxiliary fluid flow, where n is the number of macros.

Figure 15.6 1x4x3 Macros



In the **Ungrouped Macro Heat Exchanger** dialog box, in the **Geometry** tab, specify the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass**. The model will automatically extrude the macros to the depth of the heat exchanger core. For each pass, the **Number of Rows/Pass** are defined in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the direction of the auxiliary flow inlet direction and the **Number of Columns/Pass** are defined in the direction of the pass-to-pass direction.

Important

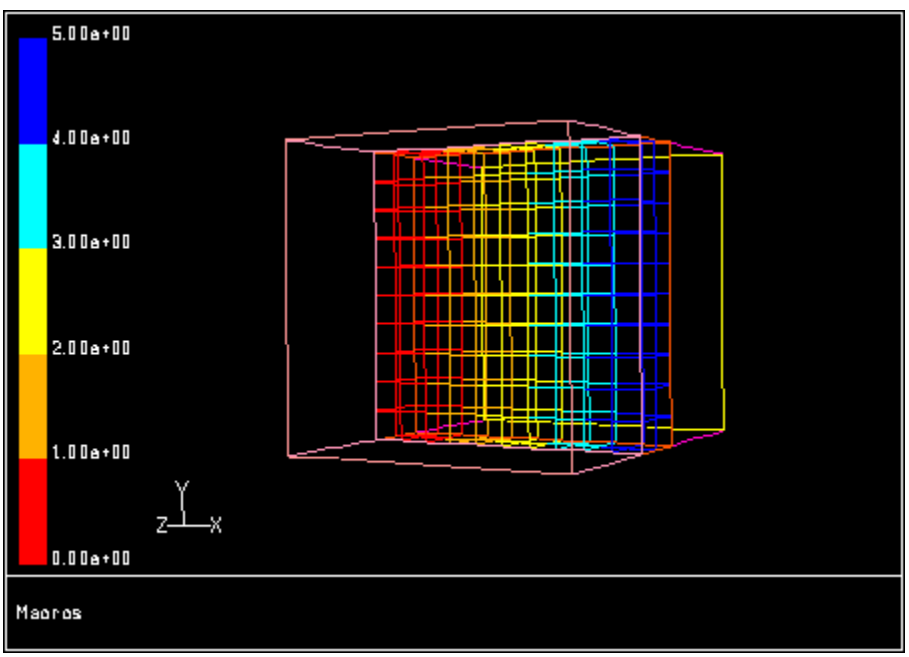
The **Number of Rows/Pass**, as well as the **Number of Columns/Pass** must be divisible by the number of cells in their respective directions. For example, if you have 50 cells in the auxiliary flow direction, you can use 25 for the **Number of Rows/Pass**, but you should not use 26 or 24. If you have 51 cells in that direction, you can only use 51 for the **Number of Rows/Pass**. The same holds true for the other direction.

15.3.4.1. Viewing the Macros

You can view the auxiliary fluid path by displaying the macros. To view the macros for your specified **Number of Passes**, **Number of Rows/Pass**, and **Number of Columns/Pass**, click the **Apply** button at the bottom of the dialog box, then click the **View Passes** button to display it. The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

For some problems, especially complex geometries, you may want to include portions of the computational-domain mesh in your macros plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the macros. This is accomplished by enabling the **Draw Mesh** option. The *Mesh Display Dialog Box* (p. 1635) will appear automatically when you enable the **Draw Mesh** option, where you can set the mesh display parameters. When you click the **View Passes** button in the **Ungrouped Macro Heat Exchanger** dialog box, the mesh display, as defined in the **Mesh Display** dialog box, will be included in the macros plot (see *Figure 15.7* (p. 778)).

Figure 15.7 Mesh Display With Macros



15.3.5. Specifying the Auxiliary Fluid Properties and Conditions

To define the auxiliary fluid properties and conditions, you will specify the **Auxiliary Fluid Flow Rate** ($\dot{m}$) in the **Auxiliary Fluid** tab. The properties of the auxiliary fluid can be specified using the **Auxiliary Fluid**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties Method drop-down list. You can choose a **constant-specific-heat** (c_p) and set the value in the **Auxiliary Fluid Specific Heat** field below, or as a user-defined function for the enthalpy using the **user-defined-enthalpy** option and selecting the corresponding UDF from the **Auxiliary Fluid Enthalpy UDF** drop-down list.

The function should return a single value depending on the index:

- Enthalpy for given values of temperature, pressure, and quality.
- Temperature for given values of enthalpy and pressure
- Specific heat for given values of temperature and pressure

The user-defined function should be of type

```
DEFINE_SOURCE(udf_name, cell_t c, Thread *t, real d[n], int index).
```

where `n` in the expression `d[n]` would be `0` for temperature, `1` for pressure, or `2` for quality. The variable `index` is `0` for enthalpy, `1` for temperature, or `2` for specific heat. This user-defined function should return

```
real value; /* (temperature or enthalpy or Cp depending on index). */
```

- If you want ANSYS FLUENT to compute the auxiliary fluid inlet temperature for a specified heat rejection, follow the steps below:
  1. Enable the **Fixed Heat Rejection** option in the **Model Data** tab.
  2. Specify the **Heat Rejection** (q_{total} in Equation 6–15 in the Theory Guide) in the **Auxiliary Fluid** tab.
  3. Specify the **Initial Temperature**, which will be used by ANSYS FLUENT as an initial guess for the inlet temperature (T_{in} in Equation 6–11 in the Theory Guide and Equation 6–16 in the Theory Guide).
- If you want ANSYS FLUENT to compute the total heat rejection of the core for a given inlet auxiliary fluid temperature, follow the steps below:
  1. Enable the **Fixed Inlet Temperature** option in the **Model Data** tab.
  2. Specify the **Inlet Temperature** (T_{in} in Equation 6–11 in the Theory Guide and Equation 6–16 in the Theory Guide) in the **Auxiliary Fluid** tab.
- If you enable the **User Defined Enthalpy** option under the **Auxiliary Fluid Properties Method**, you must also specify the **Inlet Pressure** (p_{in} in Equation 6–21 in the Theory Guide) and **Inlet Quality** (x in Equation 6–20 in the Theory Guide).

15.3.6. Setting the Pressure-Drop Parameters and Effectiveness

The pressure drop parameters and effectiveness define the **Core Porosity Model**. If you would like ANSYS FLUENT to set the porosity of this a heat exchanger zone using a particular core model, you can select the appropriate model. This will automatically set the porous media inputs. There are three ways to specify the **Core Porosity Model** parameters:

- Use the values in ANSYS FLUENT's default model.
- Define a new core porosity model with your own values.
- Read a core porosity model from an external file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you do not choose a core porosity model, you will need to set the porosity parameters in the cell zone conditions dialog box for the heat exchanger zone(s). To do this, follow the procedures described in *User Inputs for Porous Media* (p. 225).

The models you define will be saved in the case file.

15.3.6.1. Using the Default Core Porosity Model

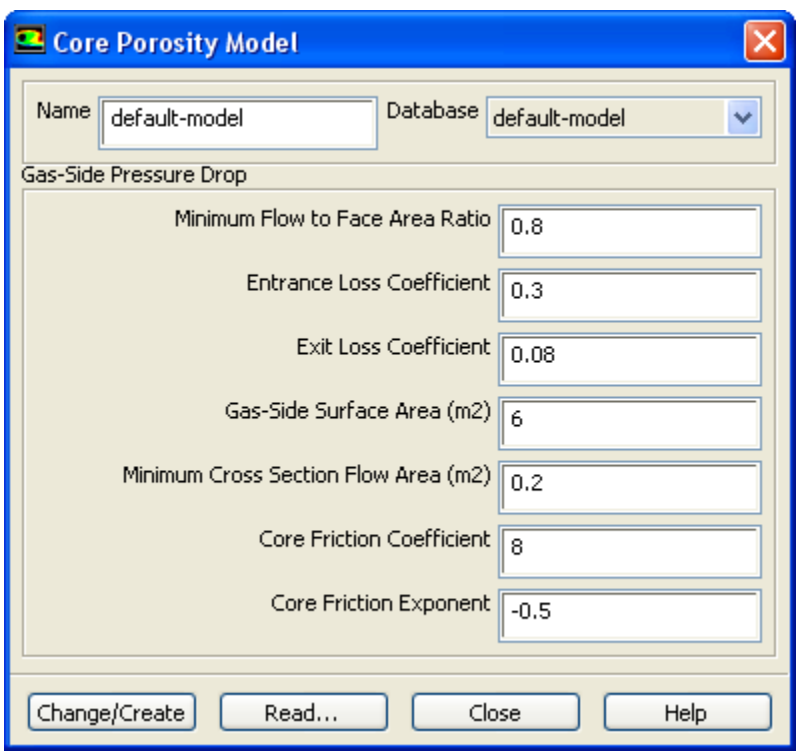
ANSYS FLUENT provides a default model for a typical heat exchanger core. The **default-model** core porosity model is a list of constant values from the **Ungrouped Macro Heat Exchanger** dialog box. These constants are used for setting the porous media parameters. To use these values, simply retain the selection of **default-model** in the **Core Porosity Model** drop-down list in the **Ungrouped Macro Heat Exchanger** dialog box. (You can view the default parameters as described below.)

15.3.6.2. Defining a New Core Porosity Model

If you want to define pressure-drop and effectiveness parameters that are different from those in the default core porosity model, you can create a new model. The steps for creating a new model are as follows:

1. Click the **Edit...** button to the right of the **Core Porosity Model** drop-down list, for which **default-model** should have been selected. This will open the *Core Porosity Model Dialog Box* (p. 1676) (*Figure 15.8* (p. 780)).

   Figure 15.8 The Core Porosity Model Dialog Box



2. Enter the name of your new model in the **Name** box at the top of the dialog box.
3. Under **Gas-Side Pressure Drop**, specify the following parameters used in Equation 6–2 in the Theory Guide:

   Minimum Flow to Face Area Ratio
   (σ)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entrance Loss Coefficient
(K_c)

Exit Loss Coefficient
(K_e)

Gas-Side Surface Area
(A)

Minimum Cross Section Flow Area
(A_c)

and the **Core Friction Coefficient** and **Core Friction Exponent** (a and b, respectively, in Equation 6–3 in the Theory Guide).

4. Click the **Change/Create** button. This will add your new model to the database.

15.3.6.3. Reading Heat Exchanger Parameters from an External File

You can read parameters for your **Core Porosity Model** from an external file. A sample file is shown below:

```
("modelname"
 (0.73 0.43 0.053 5.2 0.33 9.1 0.66))
```

The first entry in the file is the name of the model (e.g., `modelname`). The second set of numbers contains the gas-side (primary-side) pressure drop parameters:

$(\sigma\ K_c K_e\ A\ A_c\ a\ b)$

To read an external heat exchanger file, you will follow these steps:

1. In the **Core Porosity Model** dialog, click the **Read...** button.
2. In the resulting **Select File** dialog box, specify the **HXC Parameters File** name and click **OK**. ANSYS FLUENT will read the core porosity model parameters, and add the new model to the database.

15.3.6.4. Viewing the Parameters for an Existing Core Model

To view the parameters associated with a core porosity model that you have already defined, select the model name in the **Database** drop-down list (in the **Core Porosity Model** dialog box). The values for that model from the database will be displayed in the **Core Porosity Model** dialog box.

15.4. Using the Grouped Macro Heat Exchanger Model

To define a single heat exchanger that uses multiple fluid zones, or to connect the auxiliary fluid flow path among multiple heat exchangers, you can use heat exchanger groups. To use heat exchanger groups, perform the following steps:

1. Enable the calculation of energy in the *Energy Dialog Box* (p. 1645).

   Models → **Energy** → **Edit...**

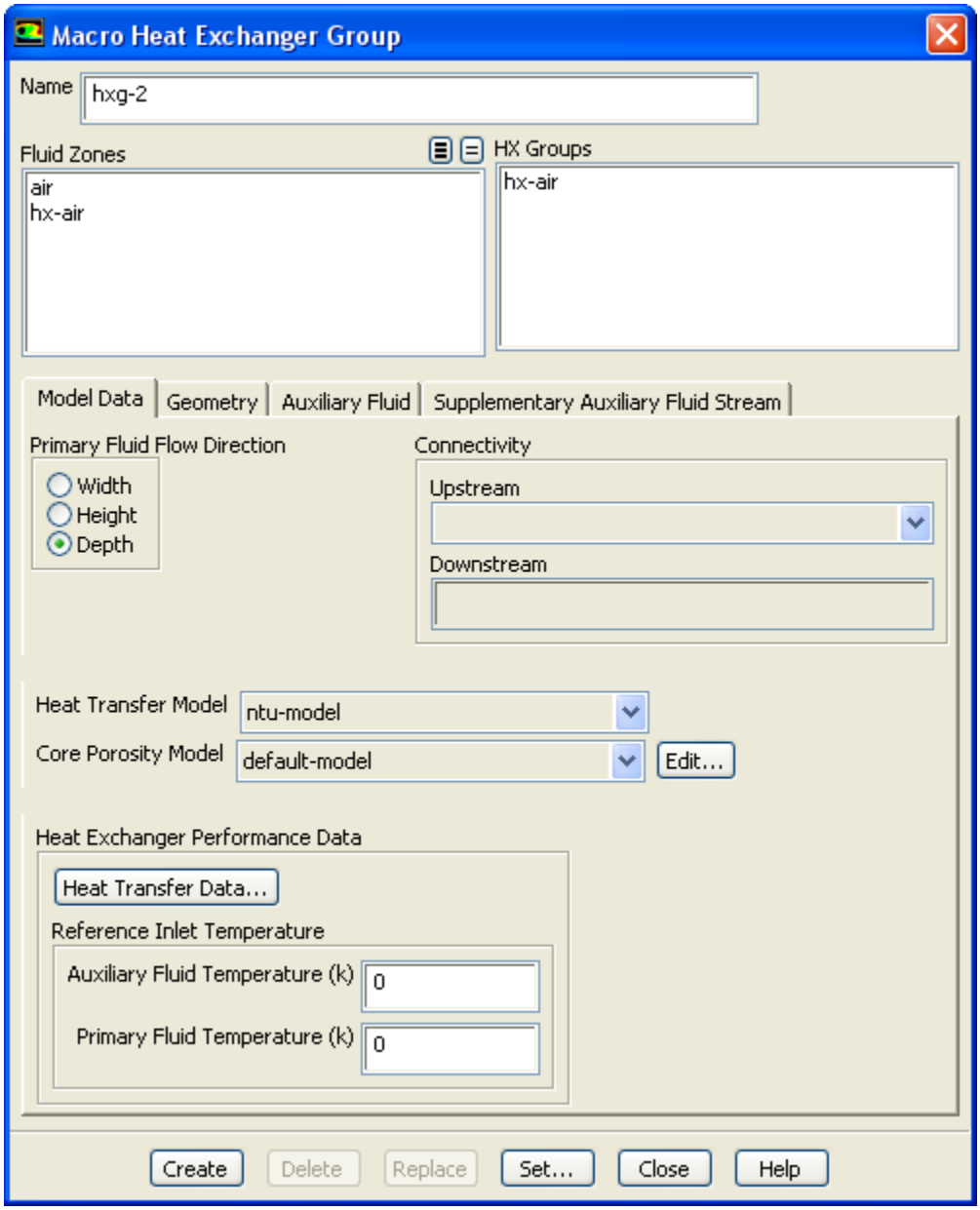
2. Enable the **Macro Model Group** option and click the **Define...** button in the *Heat Exchanger Model Dialog Box* (p. 1666) to access the **Macro Heat Exchanger Group** dialog box.

   Models → **Heat Exchanger** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the inputs to the heat exchanger group in the **Macro Heat Exchanger Group** dialog box (*Figure 15.9* (p. 782)).

Figure 15.9 The Macro Heat Exchanger Group Dialog Box



a. Enter the **Name** of the heat exchanger group.

b. Under **Fluid Zones**, select the fluid zones that you want to define in the heat exchanger group (*Selecting the Fluid Zones for the Heat Exchanger Group* (p. 786)).

c. Click the **Model Data** tab.

   i. Under **Primary Fluid Flow Direction**, specify the primary fluid flow direction as either **Width**, **Height**, or **Depth**.

   ii. Under **Connectivity**, select the **Upstream** heat exchanger group if such a connection exists (see *Selecting the Upstream Heat Exchanger Group* (p. 786)).

   iii. In the **Heat Transfer Model** drop-down list, choose either the **ntu-model** or the **simple-effectiveness-model** (see *Specifying Heat Exchanger Performance Data* (p. 776)).

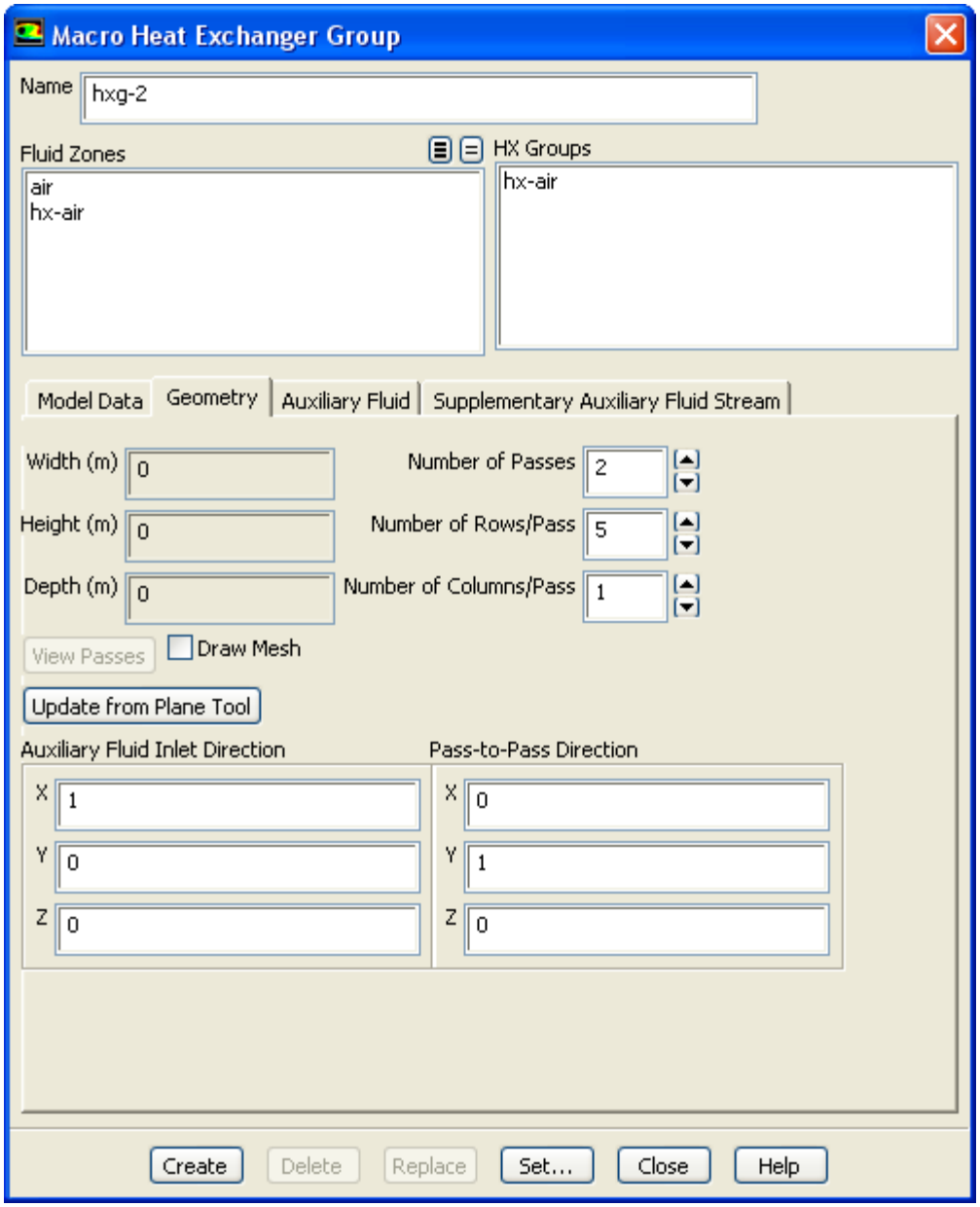
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iv. From the **Core Porosity Model** drop-down list, specify the core model that should be used to calculate the porous media parameters for the zones in the group. More information is available in *Setting the Pressure-Drop Parameters and Effectiveness* (p. 779).

v. Provide the **Heat Exchanger Performance Data** according to the chosen model.

d. Click the **Geometry** tab (*Figure 15.10* (p. 783)).

Figure 15.10 The Macro Heat Exchanger Group Dialog Box - Geometry Tab



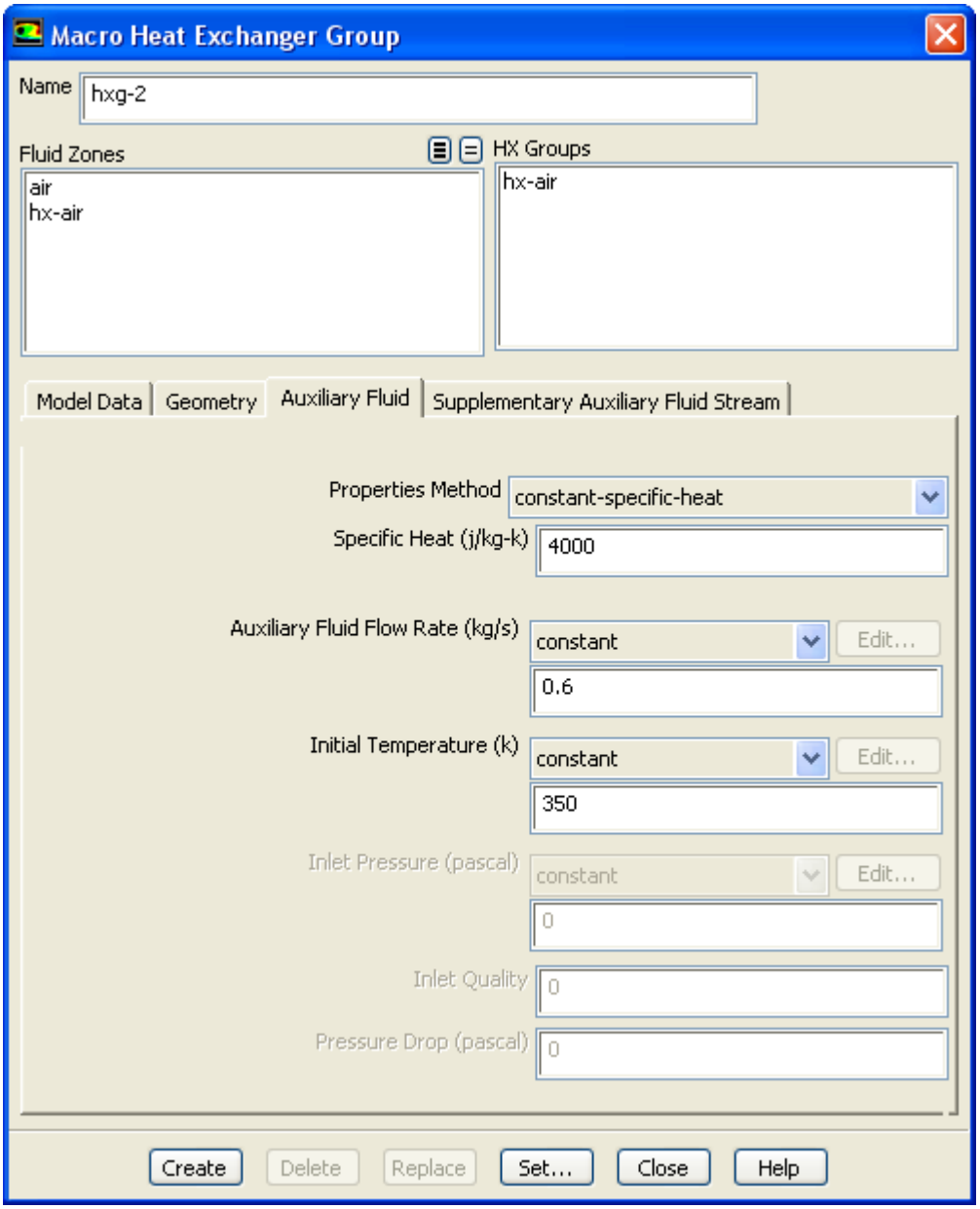
i. Define the macro mesh by specifying the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass**. More information is available in *Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions* (p. 777) and *Defining the Macros* (p. 777).

ii. Specify the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** (see *Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions* (p. 786)).

e. Click the **Auxiliary Fluid** tab (*Figure 15.11* (p. 784)) to specify the auxiliary fluid operating conditions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.11 The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab



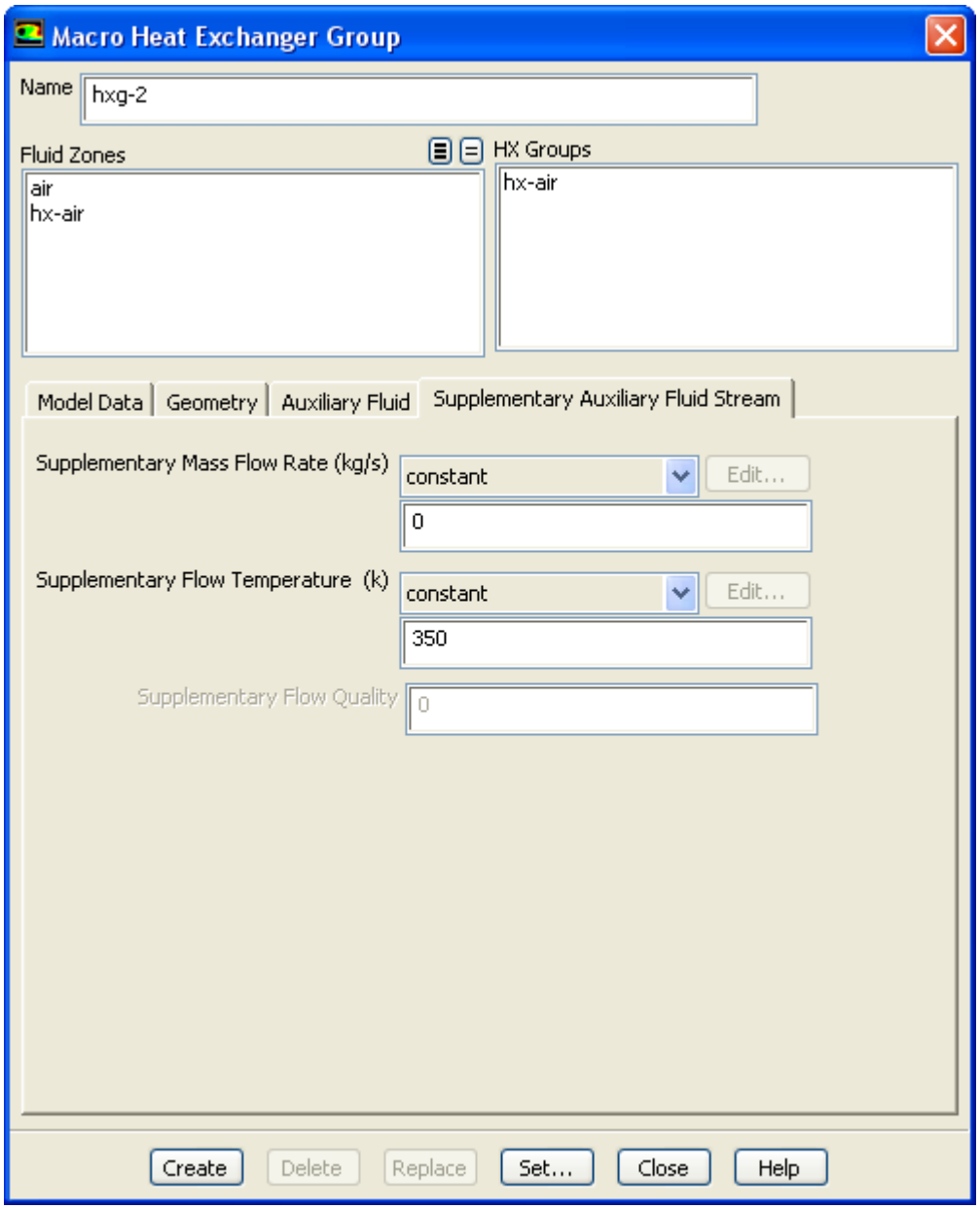
i. Specify the **Specific Heat** as either a **constant-specific-heat** or as a **user-defined-enthalpy**.

ii. **Auxiliary Fluid Flow Rate**, **Initial Temperature**, and **Inlet Pressure** can be provided as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time (see *Specifying the Auxiliary Fluid Properties and Conditions* (p. 778)).

f. If a supplementary auxiliary stream is to be modeled, click the **Supplementary Auxiliary Fluid Stream** tab.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.12 The Macro Heat Exchanger Group Dialog Box - Supplementary Auxiliary Fluid Stream Tab



i. You can specify the **Supplementary Mass Flow Rate** as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time.

ii. You can specify the **Supplementary Flow Temperature** as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time.

g. Click **Create** or **Replace** in the **Macro Heat Exchanger Group** dialog box to save all the settings. **Replace** changes the parameters of the already existing group that is selected in the **HX Groups** list.

Important

Creating or replacing any heat exchanger group initializes any previously calculated values for temperature and enthalpy for all macros.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If a heat exchanger group comprises multiple fluid zones and you wish to override any of the inputs defined in the previous steps, click the **Set...** button to open the **Ungrouped Macro Heat Exchanger** dialog box (*Figure 15.2* (p. 772)). Select the particular fluid zone as usual. Notice that the individual heat exchanger inherits the properties of the group by default. You may override any of the following:
   - **Number of Passes**, **Number of Rows/Pass**, and **Number of Columns/Pass**
   - **Auxiliary Fluid Inlet Direction** and the **Pass-to-Pass Direction**
   - **Core Porosity Model**

For additional information, please see the following sections:

15.4.1. Selecting the Fluid Zones for the Heat Exchanger Group
15.4.2. Selecting the Upstream Heat Exchanger Group
15.4.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions
15.4.4. Specifying the Auxiliary Fluid Properties
15.4.5. Specifying Supplementary Auxiliary Fluid Streams
15.4.6. Initializing the Auxiliary Fluid Temperature

15.4.1. Selecting the Fluid Zones for the Heat Exchanger Group

Select the fluid zones that you want to define in the heat exchanger group in the **Fluid Zones** drop-down list. The auxiliary fluid flow in all these zones will be in parallel. Note that one zone cannot be included in more than one heat exchanger group.

15.4.2. Selecting the Upstream Heat Exchanger Group

If you want to connect the current group in series with another group, choose the upstream heat exchanger group. Note that any group can have at most one upstream and one downstream group. Also, a group cannot be connected to itself. Select a heat exchanger group from the **Upstream** drop-down list under **Connectivity** in the **Model Data** tab of the **Macro Heat Exchanger Group** dialog box.

Important

Connecting to an upstream heat exchanger group can be done only while creating a heat exchanger group. The connection will persist even if the connection is later changed and the **Replace** button is clicked. To change a connection to an upstream heat exchanger group, you need to delete the connecting group and create a new heat exchanger group with the proper connection.

15.4.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions

The **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction**, in the **Geometry** tab can be specified as directed in *Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions* (p. 777) in the **Ungrouped Macro Heat Exchanger** dialog box. Note that the **Update from Plane Tool** will set the height, width, and depth as the average of the fluid zones selected in the **Fluid Zones**.

15.4.4. Specifying the Auxiliary Fluid Properties

The auxiliary fluid can be specified as having a **constant-specific-heat**, or a user-defined function can be written to calculate the enthalpy, as described in *Specifying the Auxiliary Fluid Properties and Conditions* (p. 778).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.5. Specifying Supplementary Auxiliary Fluid Streams

The addition or removal of a supplementary auxiliary fluid is allowed in any of the heat exchanger groups. Note that auxiliary streams are not allowed for individual zones. You will input the mass flow rate, temperature, and quality of the supplementary auxiliary fluid. You will also need to specify the heat transfer for various flow rates of primary and auxiliary flows. The auxiliary stream has the following assumptions:

- The magnitude of a negative auxiliary stream must be less than the primary auxiliary fluid inlet flow rate of the heat exchanger group.
- Added streams will be assumed to have the same fluid properties as the primary inlet auxiliary fluid.

15.4.6. Initializing the Auxiliary Fluid Temperature

When the heat exchanger group is connected to an upstream heat exchanger group, ANSYS FLUENT will automatically set the initial guess for the auxiliary fluid inlet temperature, T_{in}, to be equal to the T_{in} of the upstream heat exchanger group. Thus the boundary condition T_{in} for the first heat exchanger group in a connected series will automatically propagate as an initial guess for every other heat exchanger group in the series. However, when it is necessary to further improve convergence properties, you will be allowed to override T_{in} for any connected heat exchanger group by providing a value in the **Initial Temperature** field. Whenever such an override is supplied, ANSYS FLUENT will automatically propagate the new T_{in} to any heat exchanger groups further downstream in the series. Similarly, every time the T_{in} boundary condition for the first heat exchanger group is modified, ANSYS FLUENT will correspondingly update every downstream heat exchanger group.

If you want to impose a non-uniform initialization on the auxiliary fluid temperature field, first connect the heat exchanger groups and then set T_{in} for each heat exchanger group in streamwise order.

All heat exchangers included in a group must use the fixed T_{in} option. All heat exchangers within a heat exchanger group must have the same T_{in}. In other words, no local override of this setting is possible through the **Ungrouped Macro Heat Exchanger** dialog box.

15.5. Using the Dual Cell Heat Exchanger Model

The steps for setting up the dual cell heat exchanger model is as follows:

1. Read the mesh file containing overlapping heat exchanger cores for primary and auxiliary fluids.
2. Enable the calculation of energy in the **Energy** dialog box.

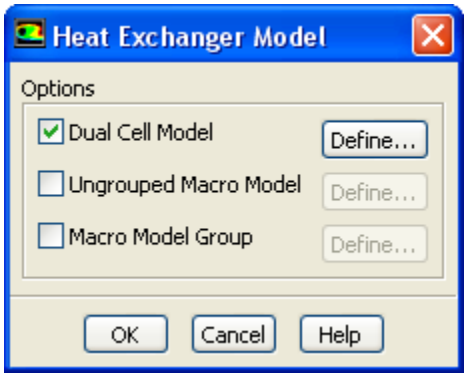
   Models → **Energy** → **Edit...**

3. Enable the **Dual Cell Model** in the **Heat Exchanger Model** dialog box and click **Define...** (*Figure 15.13* (p. 788)).

   Models → **Heat Exchanger** → **Edit...**

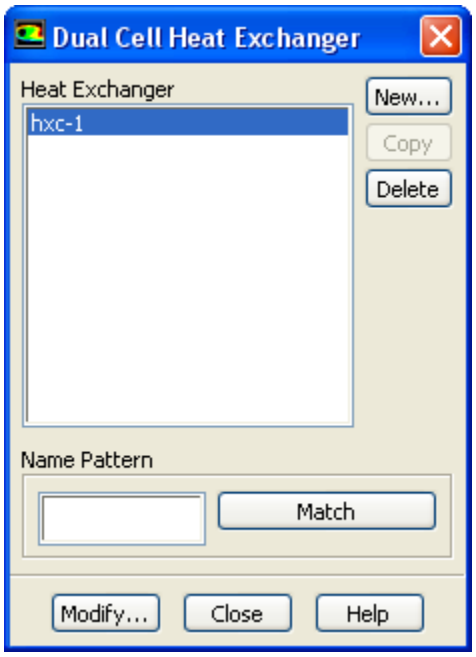
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.13 The Heat Exchanger Model Dialog Box



4. Specify the inputs to the dual cell heat exchanger model, using the **Dual Cell Heat Exchanger** dialog box (*Figure 15.14* (p. 788)).

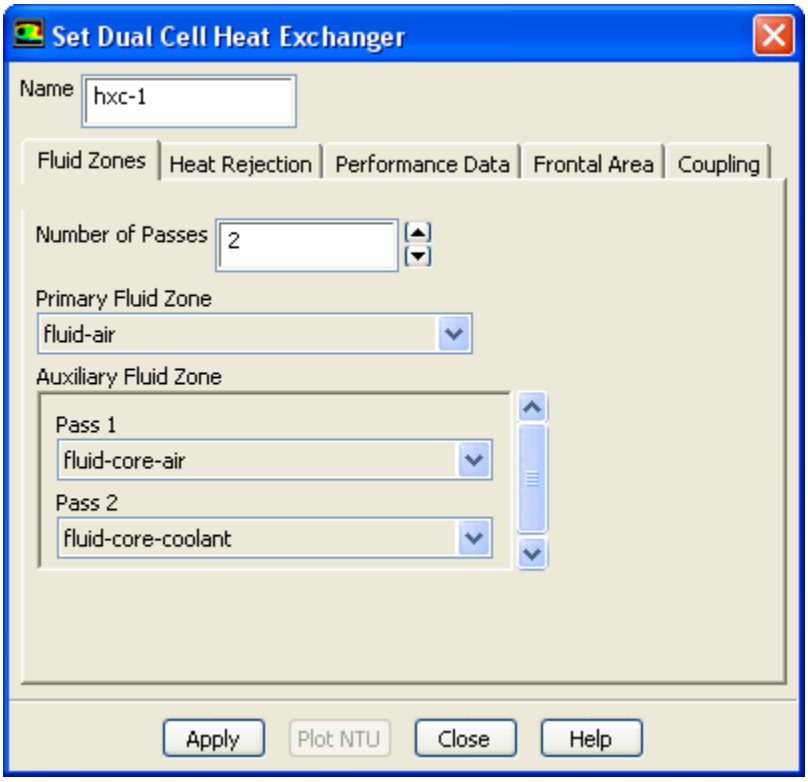
Figure 15.14 The Dual Cell Heat Exchanger Dialog Box



5. Click **New...** to define the heat exchanger. The **Set Dual Cell Heat Exchanger** dialog box will appear (*Figure 15.15* (p. 789)), where you will define the heat exchanger parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.15 The Set Dual Cell Heat Exchanger Dialog Box



a. Enter the heat exchanger **Name** or keep the default name. The suffix `-1` is incremented automatically on defining more than one heat exchanger.

b. In the **Fluid Zones** tab (*Figure 15.15* (p. 789))

   i. Specify the **Number of Passes** of your heat exchanger.

   ii. Select the appropriate **Primary**and **Auxiliary Fluid Zone**, representing the heat exchanger core.

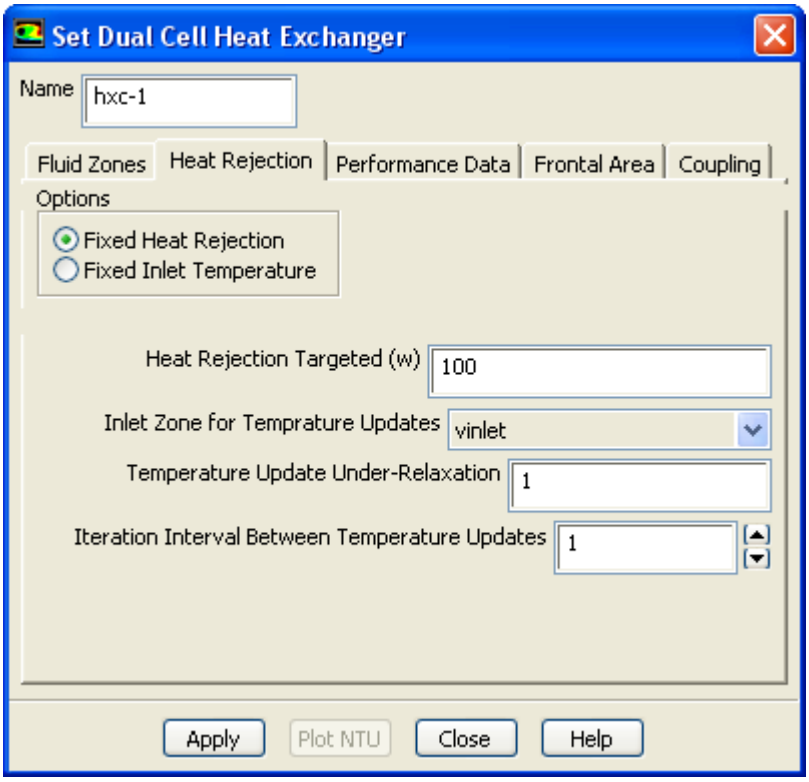
Important

The selected zones must be overlapping in physical space.

c. Click the **Heat Rejection** tab (*Figure 15.16* (p. 790)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.16 The Heat Rejection Tab



i. If you select **Fixed Heat Rejection**, set the inputs for the following:

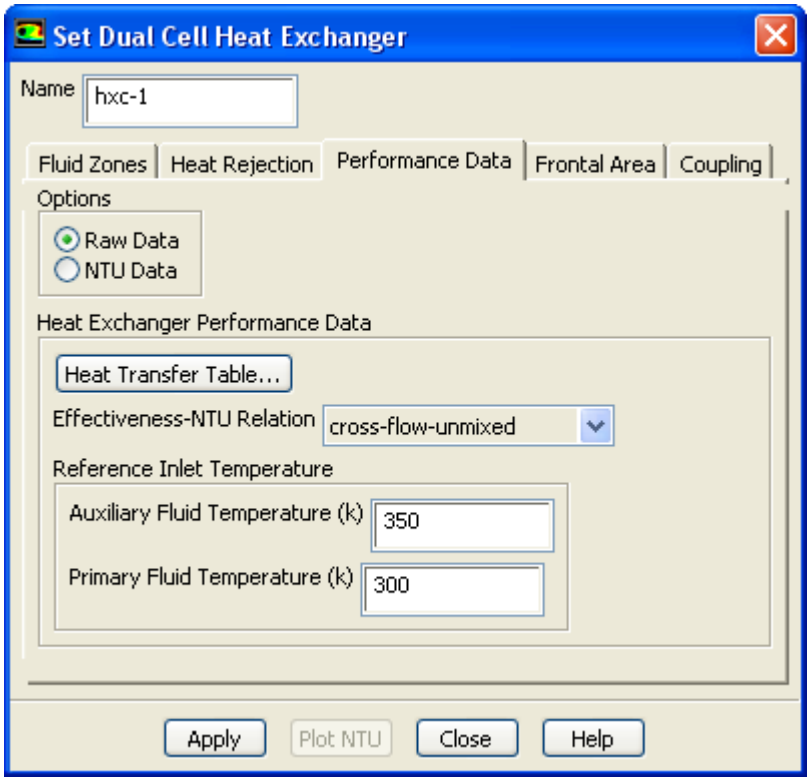
- **Heat Rejection Targeted** which is the heat rejection desired from the heat exchanger.
- **Inlet Zone for Temperature Updates** allows ANSYS FLUENT to change the temperature of the specified inlet zone in order to match the targeted heat rejection.
- **Temperature Update Under-Relaxation** is a factor which controls convergence.
- **Iteration Interval Between Temperature Updates** is used to control divergence.

ii. Select **Fixed Inlet Temperature** if the output desired is total heat rejection.

d. Click the **Performance Data** tab (*Figure 15.17* (p. 791)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.17 The Performance Data Tab



i. If you select the **Raw Data** option, then specify the following:

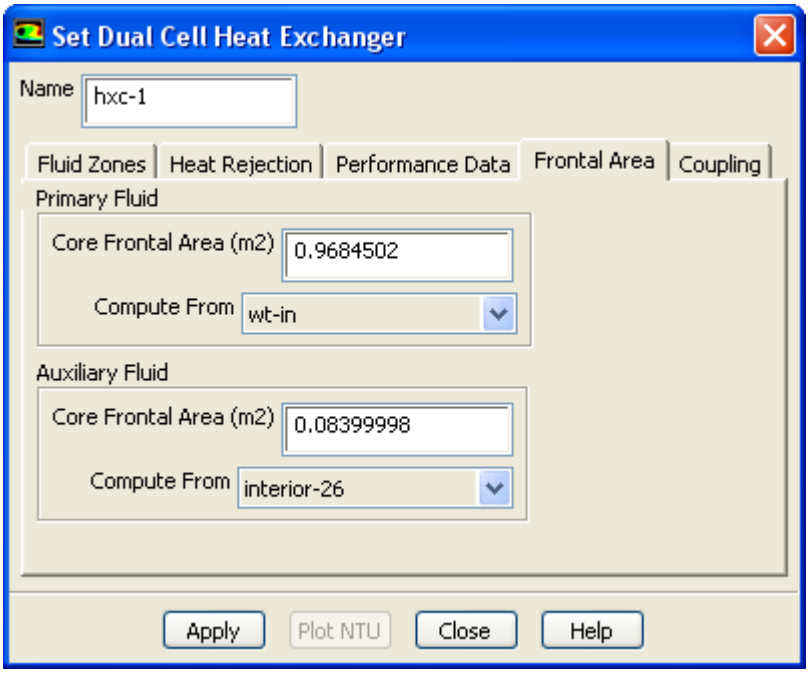
- **Heat Transfer Table...** opens the **Heat Transfer Data Table**. Populate this table as described in *Using the Ungrouped Macro Heat Exchanger Model* (p. 771).
- **Effectiveness-NTU Relation** computes the NTU values from the heat transfer data. Choose **cross-flow-unmixed**, **parallel-flow**, or **counter-flow**, all of which are described in NTU Relations.
- **Auxiliary Fluid Temperature** is the inlet reference temperature for the auxiliary fluid.
- **Primary Fluid Temperature** is the inlet reference temperature for the primary fluid.

ii. If you select the **NTU Data** option, click **NTU Table...** to access the **NTU Table** dialog box. Populate this table as described in *Using the Ungrouped Macro Heat Exchanger Model* (p. 771).

e. Click the **Frontal Area** tab. You have the option to input the **Primary** and **Auxiliary Fluid Core Frontal Area** directly, or compute the area from a surface zone, as shown in *Figure 15.18* (p. 792).

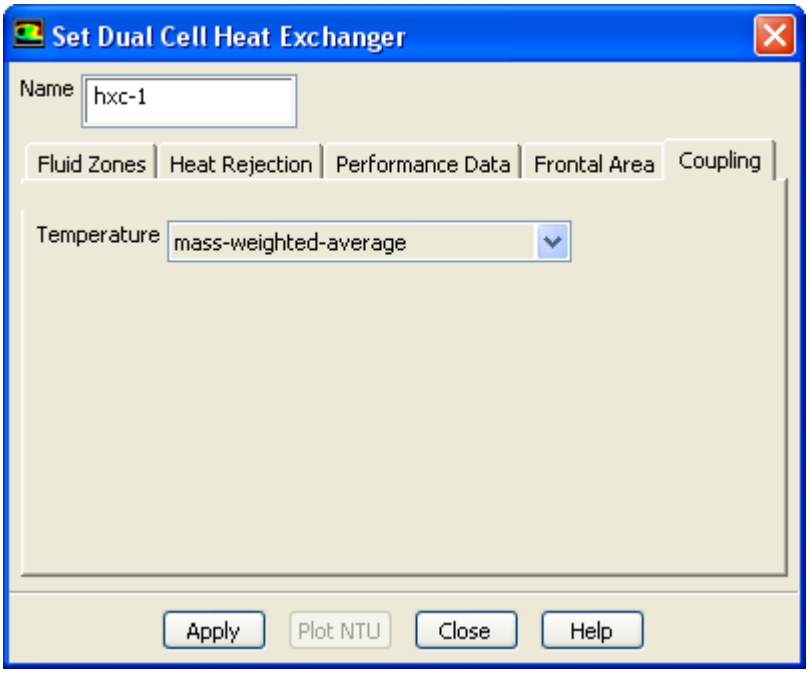
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.18 The Frontal Area Tab



f. Click the **Coupling** tab if you want to couple the heat exchanger passes (*Figure 15.19* (p. 792)).

Figure 15.19 The Coupling Tab



Consider the following example illustrating the coupling of a four-pass heat exchanger.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.20 An Example of a Four-Pass Heat Exchanger

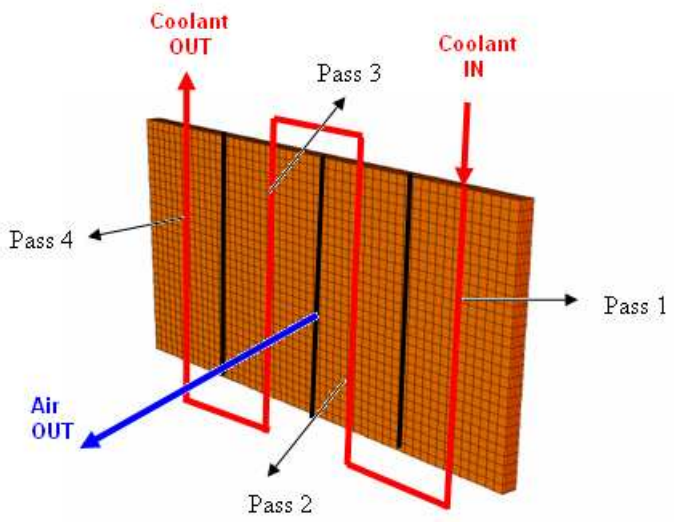


Figure 15.20 (p. 793) shows a four-pass heat exchanger with air as the primary fluid and the coolant as the auxiliary fluid. The coolant flows through the tubes in a serpentine manner and air flows normal to the tubes, forming a cross flow pattern. To model this type of flow using the dual cell heat exchanger model, you must first generate the mesh. The mesh should contain the following:

i. A single primary cell zone.

ii. Four adjacent auxiliary cell zones, one for each pass. Each auxiliary zone should be separated from the other by a coupled or uncoupled wall. Each pass will have its own inlet and outlet zones.

iii. The primary and four auxiliary zones should overlap in physical space.

In the **Coupling** tab, **mass-weighted-average** is selected by default for the **Temperature** of the outlet of Pass 1 to the inlet of Pass 2. Similarly, the mass-weighted-average temperature of the outlet of Pass 2 will be applied at the inlet zone of Pass 3, and so on. Alternatively, you can couple the passes by using **Profiles...** in the **Boundary Conditions** task page. If you do so, make sure you select **none** from the **Temperature** drop-down list in the **Coupling** tab.

Important

Make sure to specify the auxiliary zones in the correct order (i.e. the zone for Pass 1 should be selected first, then Pass 2, and so on) in the **Fluid Zones** tab of the **Set Dual Cell Heat Exchanger** dialog box.

g. Click **Apply** to save the heat exchanger inputs.

6. To view the plot of NTU Vs Primary mass flow rate for each auxiliary mass flow rate, click **Plot NTU**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Plot NTU** button will plot the performance data curve for the selected heat exchanger. The performance data is supplied through the **Performance Data** tab.

When you close the **Set Dual Cell Heat Exchanger** dialog box, you will return to the **Dual Cell Heat Exchanger** dialog box, where you should now see the heat exchanger name in the **Heat Exchanger** list.

You can

- Modify the settings of heat exchanger by selecting it from the list and clicking **Modify...**.
- Copy the data of one heat exchanger to another using the **Copy** button, assuming you have more than one heat exchanger.
- Delete any unwanted heat exchangers by selecting the heat exchanger from the list and clicking **Delete**.

Important

All the inputs are copied except for the name, primary fluid zone and auxiliary fluid zone.

15.6. Postprocessing for the Heat Exchanger Model

Postprocessing for the heat exchanger models involves computing the total heat rejection rate by setting up volume monitors and reporting of variables such as computed heat rejection, inlet or outlet temperature, specific heat, and mass flow rate.

For additional information, please see the following sections:

15.6.1. Total Heat Rejection Rate
15.6.2. Heat Exchanger Reporting

15.6.1. Total Heat Rejection Rate

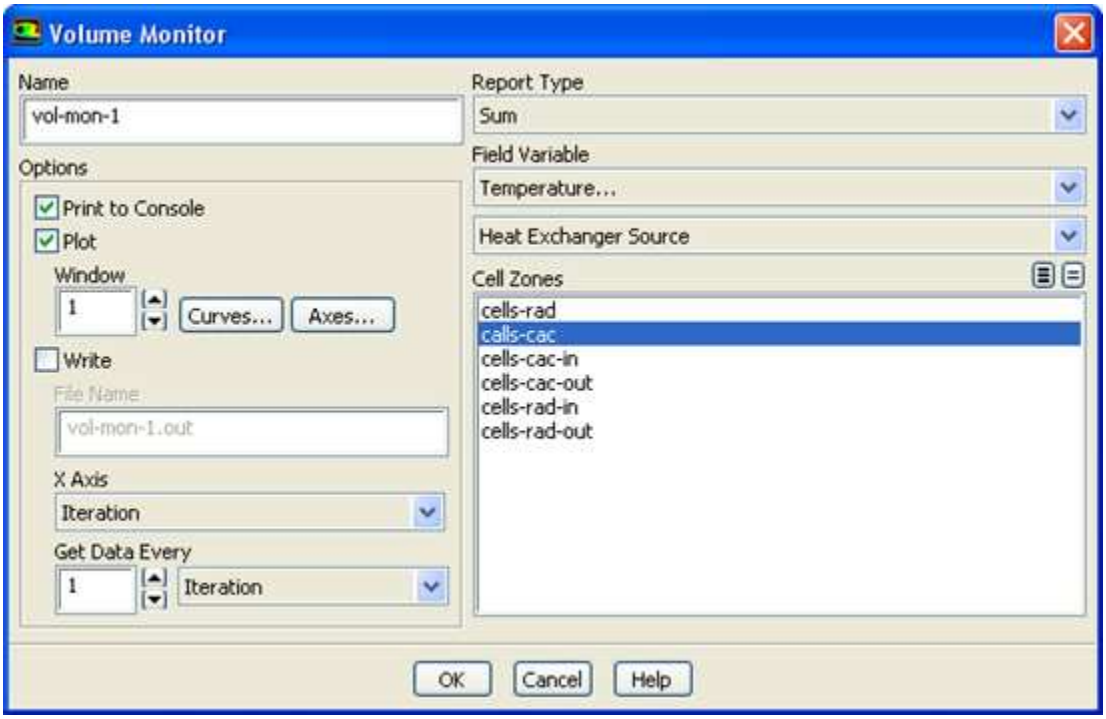
To postprocess the total heat rejection rate, you can set up a volume monitor to monitor convergence and view the computed values.

Monitors (Volume Monitors) → **Create...**

1. Select **Sum** from the **Report Type** drop-down list.
2. Select **Temperature** and **Heat Exchanger Source** from the **Field Variables** drop-down lists.
3. Select the appropriate **Cell Zones** and click **OK** to close the **Volume Monitor** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.21 The Volume Monitor Dialog Box



Important

Note that the macro heat exchangers contain only the primary fluid as the cell zone, but in case of the dual cell model, you can select either the primary or auxiliary fluid.

15.6.2. Heat Exchanger Reporting

Reporting the results for the heat exchanger models is done using the **Heat Exchanger Report** dialog box.

Reports → **Heat Exchanger** → **Set Up...**

The following variable options are available for reporting:

- **Computed Heat Rejection**
- **Inlet Temperature**
- **Outlet Temperature**
- **Mass Flow Rate**
- **Specific Heat**

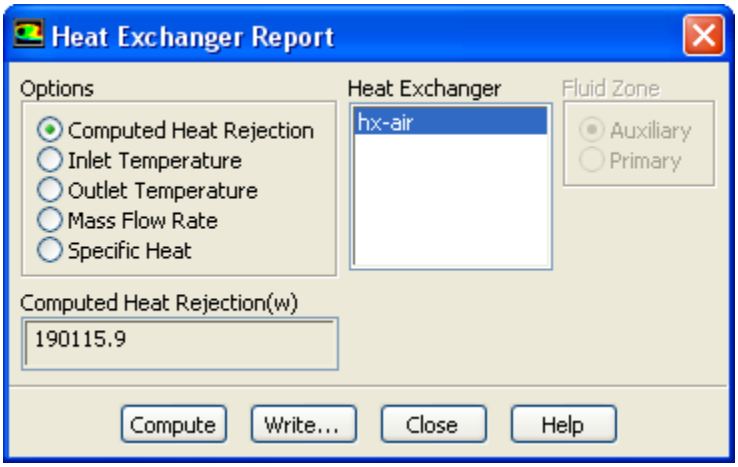
15.6.2.1. Computed Heat Rejection

To display the **Computed Heat Rejection**

1. Select **Computed Heat Rejection**from the **Options** list.
2. Select the **Heat Exchanger** from the selection list.
3. Click **Compute**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.22 The Heat Exchanger Report Dialog Box for Reporting Computed Heat Rejection

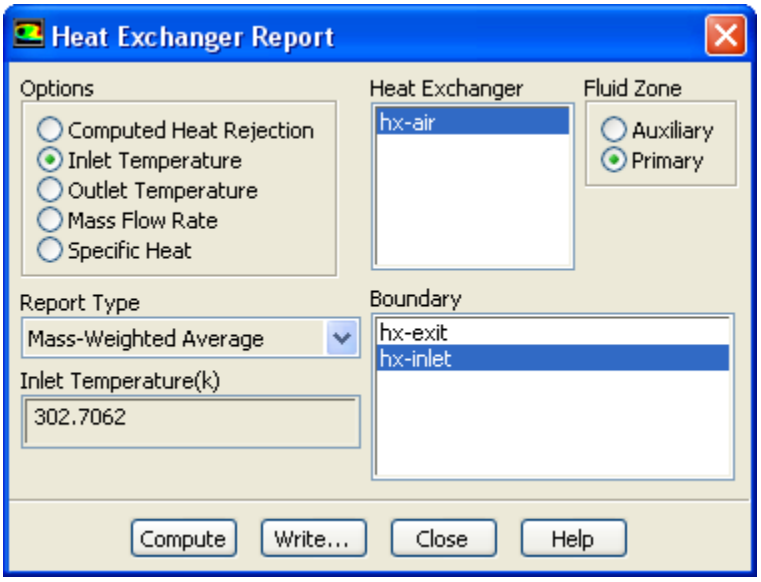


You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

15.6.2.2. Inlet/Outlet Temperature

Inlet/Outlet Temperature can be reported for both primary and auxiliary fluid in the **Heat Exchanger Report** dialog box.

Figure 15.23 The Heat Exchanger Report Dialog Box for Reporting the Inlet Temperature



1. Select **Inlet Temperature** or **Outlet Temperature** from the **Options** list.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Select the appropriate boundary zone and report type from the **Boundary** and **Report Type** lists, respectively.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the macro heat exchangers (unlike the dual cell heat exchanger) do not contain an auxiliary cell zone. Hence, the **Boundary** and **Report Type** fields will not appear in the dialog box if you are reporting an auxiliary inlet/outlet temperature.

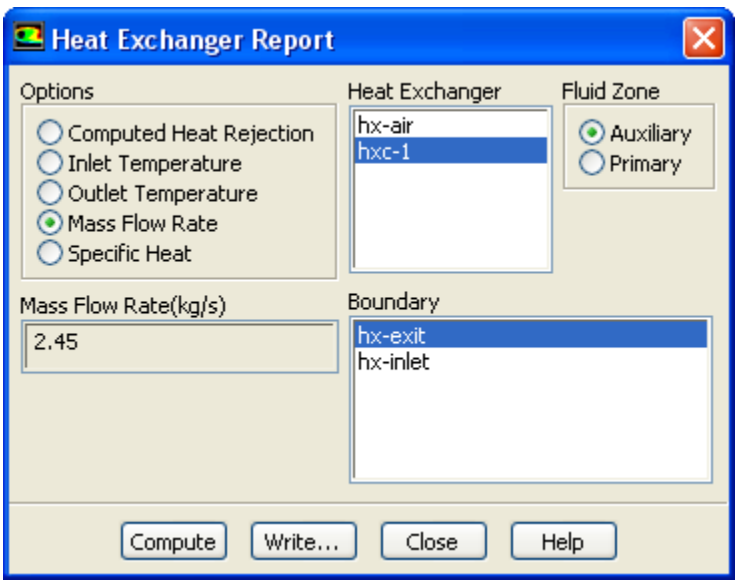
5. Click **Compute**.

You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

15.6.2.3. Mass Flow Rate

Mass Flow Rate can be reported for both the primary and auxiliary fluid in the **Heat Exchanger Report** dialog box.

Figure 15.24 The Heat Exchanger Report Dialog Box for Reporting Mass Flow Rate



1. Select **Mass Flow Rate**from the **Options** group box.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Select the appropriate boundary zone and report type from the **Boundary** and **Report Type** lists, respectively.
5. Click **Compute**.

Important

Note that the macro heat exchangers (unlike the dual cell heat exchanger) do not contain an auxiliary cell zone. Hence, the **Boundary** field will not appear in the dialog box if you are reporting an auxiliary fluid mass flow rate.

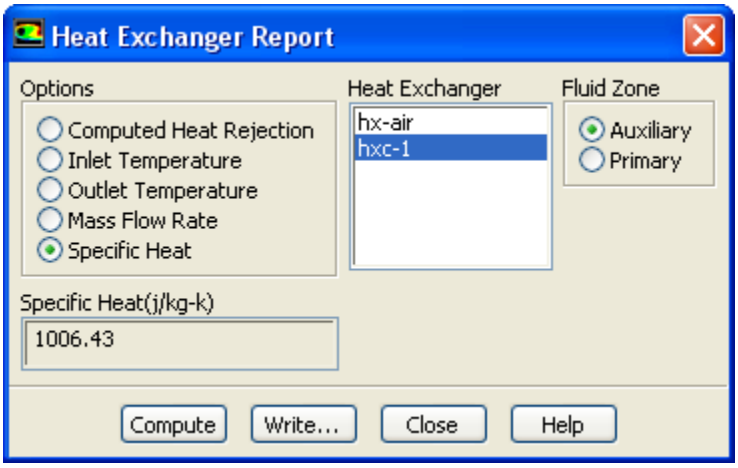
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

15.6.2.4. Specific Heat

Specific Heat for the primary or auxiliary fluid can be reported through the **Heat Exchanger Report** dialog box. If specific heat is defined as a function of temperature, the specific heat reported will be a cell volume averaged value.

Figure 15.25 The Heat Exchanger Report Dialog Box for Reporting Specific Heat



1. Select **Specific Heat** from the **Options** group box.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Click **Compute**.

15.7. Useful Reporting TUI Commands

To report the results for the macro heat exchangers, you can use the following text command:

`define` → `models` → `heat-exchanger` → `macro-model` → `heat-exchanger-macro-report`

Specify the fluid zone *id/name* for which you want to obtain information.

To view the connectivity of the heat exchanger groups, use the text command:

```
(report-connectivity)
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 16: Modeling Species Transport and Finite-Rate Chemistry

ANSYS FLUENT can model the mixing and transport of chemical species by solving conservation equations describing convection, diffusion, and reaction sources for each component specie. Multiple simultaneous chemical reactions can be modeled, with reactions occurring in the bulk phase (volumetric reactions) and/or on wall or particle surfaces, and in the porous region. Species transport modeling capabilities, both with and without reactions, and the inputs you provide when using the model are described in this chapter. For theoretical information about species transport, see "Species Transport and Finite-Rate Chemistry" in the Theory Guide.

Note that you may also want to consider modeling your turbulent reacting flame using the mixture fraction approach (for non-premixed systems, described in *Modeling Non-Premixed Combustion* (p. 839)), the reaction progress variable approach (for premixed systems, described in *Modeling Premixed Combustion* (p. 893)), the partially premixed approach (described in *Modeling Partially Premixed Combustion* (p. 905)), or the composition PDF Transport approach (described in *Modeling a Composition PDF Transport Problem* (p. 911)). Modeling multiphase species transport and finite-rate chemistry can be found in *Modeling Multiphase Flows* (p. 1091).

Information is divided into the following sections:

16.1. Volumetric Reactions
16.2. Wall Surface Reactions and Chemical Vapor Deposition
16.3. Particle Surface Reactions
16.4. Species Transport Without Reactions

16.1. Volumetric Reactions

Information about using species transport and finite-rate chemistry as related to volumetric reactions is presented in the following subsections. For more information about the theoretical background of volumetric reactions, see Volumetric Reactions in the Theory Guide.

16.1.1. Overview of User Inputs for Modeling Species Transport and Reactions
16.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material
16.1.3. Defining Properties for the Mixture and Its Constituent Species
16.1.4. Setting up Coal Simulations with the Coal Calculator Dialog Box
16.1.5. Defining Cell Zone and Boundary Conditions for Species
16.1.6. Defining Other Sources of Chemical Species
16.1.7. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry
16.1.8. Postprocessing for Species Calculations
16.1.9. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format

16.1.1. Overview of User Inputs for Modeling Species Transport and Reactions

The basic steps for setting up a problem involving species transport and reactions are listed below, and the details about performing each step are presented in *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801) – *Defining Cell Zone and Boundary Conditions for Species* (p. 822). Additional information about setting up and solving the problem is provided in *Defining Other Sources of Chemical Species* (p. 822) – *Postprocessing for Species Calculations* (p. 826).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable species transport and volumetric reactions, and specify the mixture material. See *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801). The mixture material concept is explained in *Mixture Materials* (p. 800).
2. If you also modeling wall or particle surface reactions, enable wall surface and/or particle surface reactions. See *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829) *Particle Surface Reactions* (p. 835) for details.
3. Check and/or define the properties of the mixture. See *Defining Properties for the Mixture and Its Constituent Species* (p. 805). Mixture properties include the following:
   - species in the mixture
   - reactions
   - other physical properties (e.g., viscosity, specific heat)
4. Check and/or set the properties of the individual species in the mixture. See *Defining Properties for the Mixture and Its Constituent Species* (p. 805).
5. Set speciescell zone and boundary conditions. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822).

In many cases, you will not need to modify any physical properties because the solver gets species properties, reactions, etc. from the materials database when you choose the mixture material. Some properties, however, may not be defined in the database. You will be warned when you choose your material if any required properties need to be set, and you can then assign appropriate values for these properties. You may also want to check the database values of other properties to be sure that they are correct for your particular application. For details about modifying an existing mixture material or creating a new one from scratch, see *Defining Properties for the Mixture and Its Constituent Species* (p. 805). Modifications to the mixture material can include the following:

- Addition or removal of species
- Changing the chemical reactions
- Modifying other material properties for the mixture
- Modifying material properties for the mixture's constituent species

If you are solving a reacting flow, you will usually want to define the mixture's specific heat as a function of composition, and the specific heat of each species as a function of temperature. You may want to do the same for other properties as well. By default, most species specific heats in the database are piecewise-polynomial functions of temperature, but you may choose to specify a different temperature-dependent function if you know of one that is more suitable for your problem.

16.1.1.1. Mixture Materials

The concept of mixture materials has been implemented in ANSYS FLUENT to facilitate the setup of species transport and reacting flow. A mixture material may be thought of as a set of species and a list of rules governing their interaction. The mixture material carries with it the following information:

- A list of the constituent species, referred to as "fluid" materials
- A list of mixing laws dictating how mixture properties (density, viscosity, specific heat, etc.) are to be derived from the properties of individual species if composition-dependent properties are desired
- A direct specification of mixture properties if composition-independent properties are desired
- Diffusion coefficients for individual species in the mixture

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Other material properties (e.g., absorption and scattering coefficients) that are not associated with individual species
- A set of reactions, including a reaction type (finite-rate, eddy-dissipation, etc.) and stoichiometry and rate constants

Both mixture materials are stored in the ANSYS FLUENT materials database. Many common mixture materials are included (e.g., methane-air, propane-air). Generally, one/two-step reaction mechanisms and many physical properties of the mixture and its constituent species are defined in the database. When you indicate which mixture material you want to use, the appropriate mixture material, fluid materials, and properties are loaded into the solver. If any necessary information about the selected material (or the constituent fluid materials) is missing, the solver will inform you that you need to specify it. In addition, you may choose to modify any of the predefined properties. See *Using the Materials Task Page* (p. 387) for information about the sources of ANSYS FLUENT's database property data.

For example, if you plan to model combustion of a methane-air mixture, you do not need to explicitly specify the species involved in the reaction or the reaction itself. You will simply select **methane-air** as the mixture material to be used, and the relevant species (CH_4, O_2, CO_2, H_2O, and N_2) and reaction data will be loaded into the solver from the database. You can then check the species, reactions, and other properties and define any properties that are missing and/or modify any properties for which you wish to use different values or functions. You will generally want to define a composition- and temperature-dependent specific heat, and you may want to define additional properties as functions of temperature and/or composition.

The use of mixture materials is performed in the *Create/Edit Materials Dialog Box* (p. 1736), as described in *Defining Properties for the Mixture and Its Constituent Species* (p. 805).

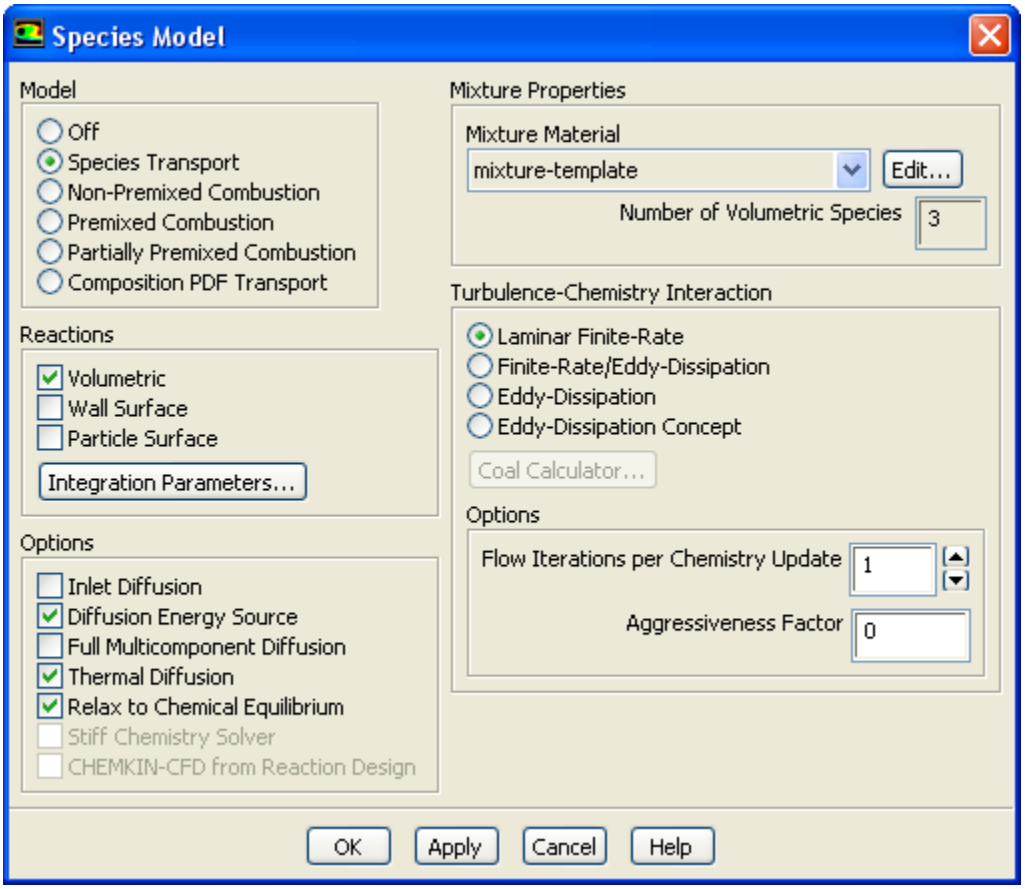
16.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material

The problem setup for species transport and volumetric reactions begins in the **Species Model** dialog box (*Figure 16.1* (p. 802)). For cases which involve multiphase species transport and reactions, refer to Modeling Species Transport in Multiphase Flows in the Theory Guide.

Models → **Species** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.1 The Species Model Dialog Box



1. Under **Model**, select **Species Transport**.
2. Under **Reactions**, enable **Volumetric**.
3. In the **Mixture Material** drop-down list under **Mixture Properties**, choose which mixture material you want to use in your problem. The drop-down list will include all of the mixtures that are currently defined in the database. To check the properties of a mixture material, select it and click the **Edit...** button. If the mixture you want to use is not in the list, choose the **mixture-template** material, and see *Defining Properties for the Mixture and Its Constituent Species* (p. 805) for details on setting your mixture's properties. If there is a mixture material listed that is similar to your desired mixture, you may choose that material and see *Defining Properties for the Mixture and Its Constituent Species* (p. 805) for details on modifying properties of an existing material.

   When you choose the **Mixture Material**, the **Number of Volumetric Species** in the mixture will be displayed in the dialog box for your information.

Important

Note that if you re-open the **Species Model** dialog box after species transport has already been enabled, only the mixture materials available in your case will appear in the list. You can add more mixture materials to your case by copying them from the database, as described in *Copying Materials from the ANSYS FLUENT Database* (p. 389), or by creating a new mixture, as described in *Creating a New Material* (p. 390) and *Defining Properties for the Mixture and Its Constituent Species* (p. 805).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As mentioned in *Mixture Materials* (p. 800), modeling parameters for the species transport and (if relevant) reactions will automatically be loaded from the database. If any information is missing, you will be informed of this after you click **OK** in the **Species Model** dialog box. If you want to check or modify any properties of the mixture material, you will use the *Create/Edit Materials Dialog Box* (p. 1736), as described in *Defining Properties for the Mixture and Its Constituent Species* (p. 805).

4. Choose the **Turbulence-Chemistry Interaction** model. Four models are available:

   Laminar Finite-Rate
   computes only the Arrhenius rate (see Equation 7–8 in the Theory Guide) and neglects turbulence-chemistry interaction. You can specify the following inputs:

   Flow Iterations per Chemistry Update
   Increasing the number reduces the computational expense of the chemistry calculations. By default, ANSYS FLUENT will update the chemistry once per 10 flow iterations. This option is available when either the **Relax to Chemical Equilibrium** or the **Stiff Chemistry Solver** option is enabled.

   Aggressiveness Factor
   This is a numerical factor which controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 (the default) is the most robust, but results in the slowest convergence. This option is available when either the **Relax to Chemical Equilibrium** or the **Stiff Chemistry Solver** option is enabled.

   Finite-Rate/Eddy-Dissipation
   (for turbulent flows only) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two. You can specify the **Flow Iterations per Chemistry Update** and **Aggressiveness Factor** if you have the **Relax to Chemical Equilibrium** option enabled.

   Eddy-Dissipation
   (for turbulent flows only) computes only the mixing rate (see Equation 7–24 and Equation 7–25 in the Theory Guide). You can specify the **Flow Iterations per Chemistry Update** and **Aggressiveness Factor** if you have the **Relax to Chemical Equilibrium** option enabled.

   Eddy-Dissipation Concept
   (for turbulent flows only) models turbulence-chemistry interaction with detailed chemical mechanisms (see Equation 7–8 Equation 7–29 in the Theory Guide). When using this model, you can modify the following:

   Flow Iterations per Chemistry Update
   Increasing the number reduces the computational expense of the chemistry calculations. By default, ANSYS FLUENT will update the chemistry once per 10 flow iterations.

   Aggressiveness Factor
   This is a numerical factor which controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 (the default) is the most robust, but results in the slowest convergence.

   Volume Fraction Constant
   and the **Time Scale Constant** (C_ξ in Equation 7–27 and C_τ in Equation 7–28 in the Theory Guide), although the default values are recommended.

5. You can set the integration parameters for the **Laminar Finite-Rate** and **Eddy-Dissipation Concept** models by clicking the **Integration Parameters...** button under **Reactions**. When using ISAT for chemistry tabulation, it is important to set appropriate maximum table size and error tolerance. For details about this option, see *Steps for Using the Composition PDF Transport Model* (p. 911).

6. (optional) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See *Full Multicomponent Diffusion* (p. 440) for details.

7. (optional) If you want to model Relaxation to Chemical Equilibrium or include the stiff chemistry solver, enable the **Relax to Chemical Equilibrium** or **Stiff Chemistry Solver** option. For information about the Relaxation to Chemical Equilibrium model, please refer to The Relaxation to Chemical Equilibrium Model in the Theory Guide.

Important

The **Relax to Chemical Equilibrium** model requires thermodynamic data of all the species to calculate chemical equilibrium. This thermodynamic data is contained (by default) in the file `.../fluent13.x/cpropep/data/thermo.db`. If the user defines his or her own species in ANSYS FLUENT, as determined by the **Chemical Formula** in the **Create/Edit Materials** dialog box, and this species name is not in the thermo.db file, ANSYS FLUENT will report an error. To overcome this error, the you must manually enter all the unknown species in the thermo file.

8. Enabling **CHEMKIN-CFD from Reaction Design** for laminar reactions, will allow you to use the proprietary reaction-rate utilities and solution algorithms from Reaction Design, which is based on and compatible with their CHEMKIN technology [*41*] (p. 2211). For **Eddy-Dissipation Concept**, **Turbulence-Chemistry Interaction** and the **Composition PDF Transport** model, enabling the **CHEMKIN-CFD from Reaction Design** option will allow you to use reaction rates from Reaction Design's KINetics module, instead of the default ANSYS FLUENT reaction rates. ANSYS FLUENT's ISAT algorithm is employed to integrate these rates.

   Refer to the KINetics for Fluent manual [*2*] (p. 2209) from Reaction Design for details on the chemistry formulation options. For more information, or to obtain a license to the Fluent/KINetics module, please contact Reaction Design at `info@reactiondesign.com` info or +1 858-550-1920, or go to `www.reactiondesign.com`.

9. Enable the **Thickened Flame Model** to model laminar flames, or more typically as an LES combustion model for turbulent premixed and partially-premixed flame. Specify the **Number of Grid Points in Flame**, which by default are 8 grid points. Consequently, in the **Create/Edit Materials** dialog box, you will specify the **Laminar Flame Speed**, for which you have a choice of **constant**, **user-defined**, or **metghalchi-keck-law**, and the **Laminar Flame Thickness**, for which you have a choice of **constant**, **user-defined**, or **diffusivity-over-flame-speed**.

Note

When the **Thickened Flame Model** is enabled (*Figure 16.2* (p. 805)), you have the option to hook a UDF in the **User-Defined Function Hooks** dialog box to customize the **Thickened Flame Model** parameters. Refer to DEFINE_THICKENED_FLAME_MODEL in the UDF manual for details.

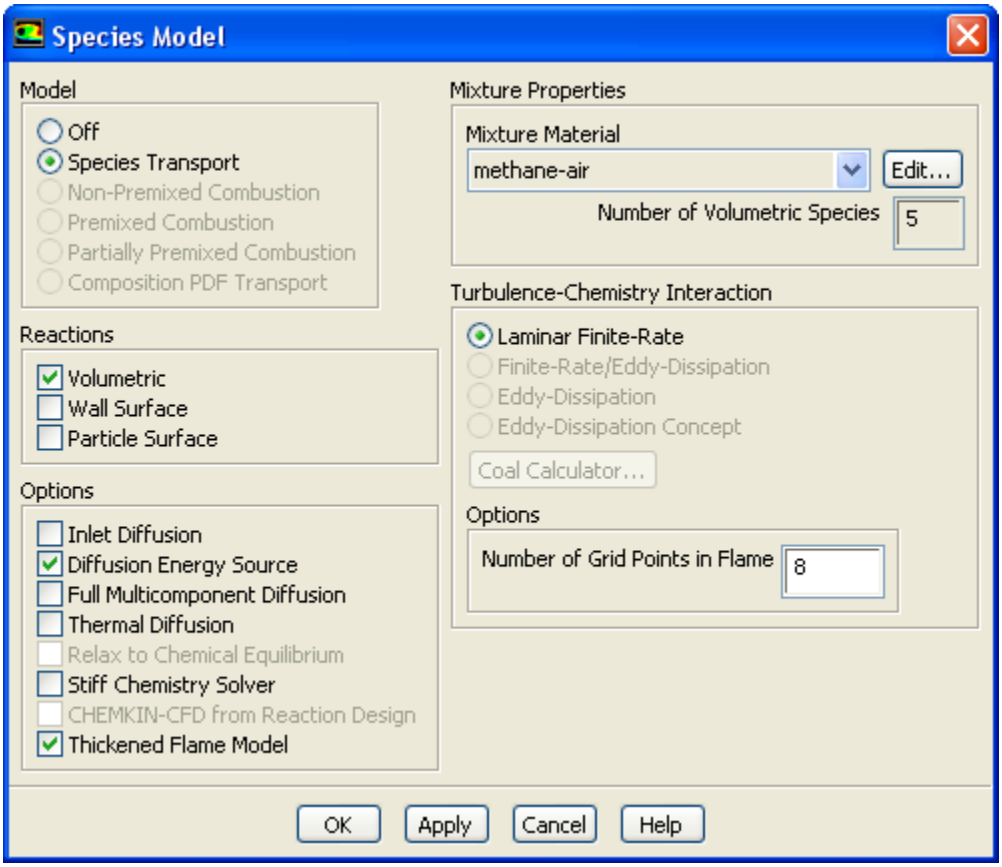
Important

The **Thickened Flame Model** is available only for unsteady laminar or turbulent (LES/DES/SAS) flows, with **Species Transport** and **Volumetric Reactions** enabled.

For information about the theory behind the **Thickened Flame Model**, refer to The Thickened Flame Model in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.2 The Species Model Dialog Box Displaying the Thickened Flame Model



16.1.3. Defining Properties for the Mixture and Its Constituent Species

As discussed in *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799), if you use a mixture material from the database, most mixture and species properties will already be defined. You may follow the procedures in this section to check the current properties, modify some of the properties, or set all properties for a brand-new mixture material that you are defining from scratch.

Remember that you will need to define properties for the mixture material and also for its constituent species. It is important that you define the mixture properties before setting any properties for the constituent species, since the species property inputs may depend on the methods you use to define the properties of the mixture. The recommended sequence for property inputs is as follows:

1. Define the mixture species, and reaction(s), and define physical properties for the mixture. Remember to click the **Change/Create** button when you are done setting properties for the mixture material.
2. Define physical properties for the species in the mixture. Remember to click the **Change/Create** button after defining the properties for each specie.

These steps, all of which are performed in the *Create/Edit Materials Dialog Box* (p. 1736), are described in detail in this section.

Materials

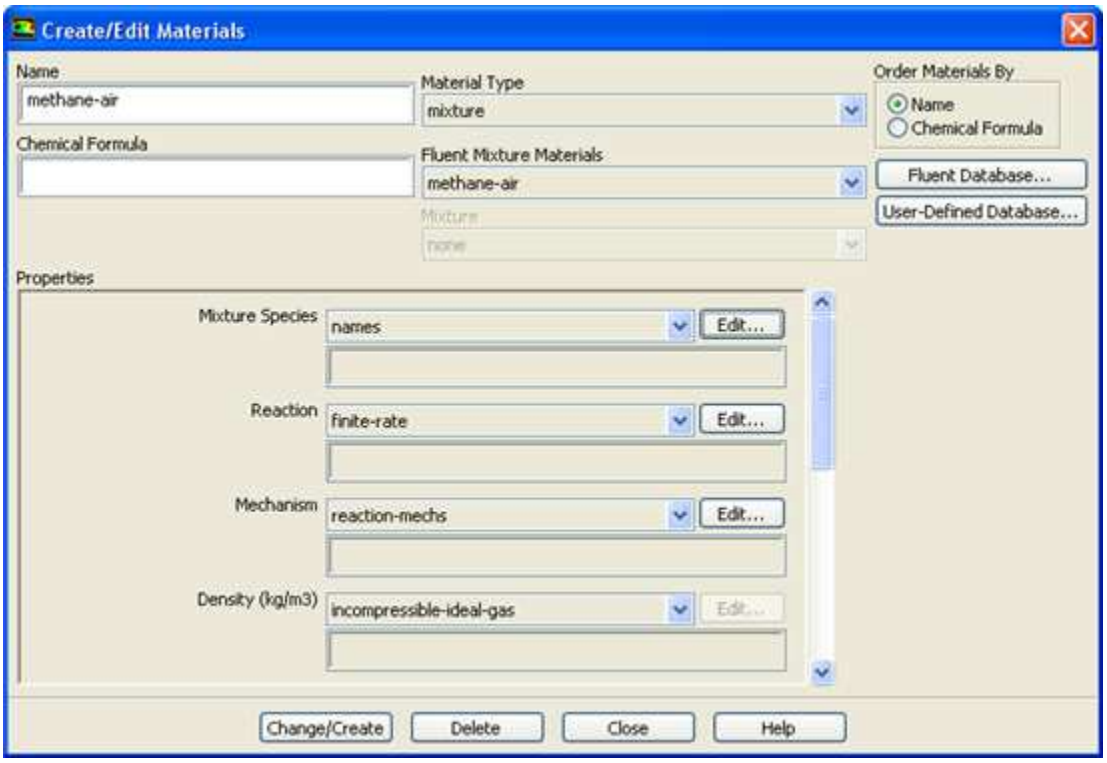
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.1.3.1. Defining the Species in the Mixture

If you are using a mixture material from the database, the species in the mixture will already be defined for you. If you are creating your own material or modifying the species in an existing material, you will need to define them yourself.

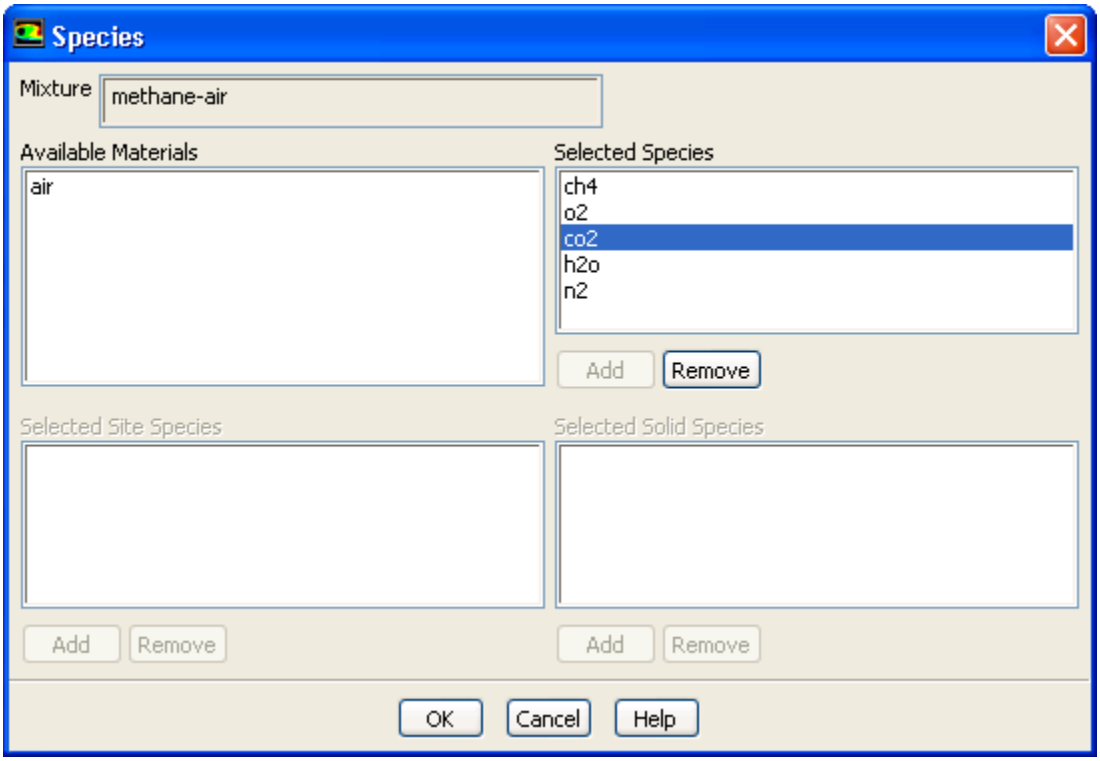
In the **Create/Edit Materials** dialog box (*Figure 16.3* (p. 806)), check that the **Material Type** is set to **mixture** and your mixture is selected in the **Fluent Mixture Materials** list. Click the **Edit...** button to the right of **Mixture Species** to open the *Species Dialog Box* (p. 1763) (*Figure 16.4* (p. 807)).

Figure 16.3 The Create/Edit Materials Dialog Box (Showing a Mixture Material)



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.4 The Species Dialog Box



16.1.3.1.1. Overview of the Species Dialog Box

In the **Species** dialog box, the **Selected Species** list shows all of the fluid-phase species in the mixture. If you are modeling wall or particle surface reactions, the **Selected Solid Species** list will show all of the bulk solid species in the mixture. Solid species are species that are deposit to, or etch from, wall boundaries or discrete-phase particles (e.g., Si(s)) and do not exist as fluid-phase species. If you are modeling wall surface reactions with site balancing, where species adsorb onto the wall surface, react, and then desorb off the surface, the **Selected Site Species** list will show all of the site species in the mixture.

The use of solid and site species with wall surface reactions is described in *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829). See *Particle Surface Reactions* (p. 835) for information about particle surface reactions.

Important

The order of the species in the **Selected Species** list is very important. ANSYS FLUENT considers the last species in the list to be the bulk specie. You should therefore be careful to retain the most abundant species (by mass) as the last species when you add species to or delete species from a mixture material.

The **Available Materials** list shows materials that are available but not in the mixture. Generally, you will see air in this list, since air is always available by default.

16.1.3.1.2. Adding Species to the Mixture

If you are creating a mixture from scratch or starting from an existing mixture and adding some missing species, you will first need to load the desired species from the database (or create them, if they are not present in the database) so that they will be available to the solver. You will need to close the **Species** dialog

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box before you begin, since it is a "modal" dialog box that will not allow you to do anything else when it is open. The procedure for adding species is as follows:

1. In the **Create/Edit Materials** dialog box, click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box and copy the desired specie, as described in *Copying Materials from the ANSYS FLUENT Database* (p. 389). Remember that the constituent species of the mixture are fluid materials, so you should select **fluid** as the **Material Type** in the **Fluent Database Materials** dialog box to see the correct list of choices. Note that available solid and site species (for surface reactions) are also contained in the **fluid** list.

   Important

   If you do not see the species you are looking for in the database, you can create a new fluid material for that specie, following the instructions in *Creating a New Material* (p. 390), and then continue with step 2, below.

2. Re-open the **Species** dialog box. You will see that the fluid materials you copied from the database (or created) are listed in the **Available Materials** list.
3. To add a species to the mixture, select it in the **Available Materials** list and click the **Add** button below the **Selected Species** list (or below the **Selected Site Species** or **Selected Solid Species** list, to define a site or solid species). The species will be added to the end of the relevant list and removed from the **Available Materials** list.
4. Repeat the previous step for all the desired species. When you are finished, click the **OK** button.

Important

Adding a species to the list will alter the order of the species. You should be sure that the last species in the list is the bulk specie, and you should check all cell and boundary zone conditions, under-relaxation factors, and other solution parameters that you have set, as described in detail in the following sections.

16.1.3.1.3. Removing Species from the Mixture

To remove a species from the mixture, simply select it in the **Selected Species** list (or the **Selected Site Species** or **Selected Solid Species** list) and click the **Remove** button below the list. The species will be removed from the list and added to the **Available Materials** list.

Important

Removing a species from the list will alter the order of the species. You should be sure that the last species in the list is the bulk species, and you should check any cell zone or boundary conditions, under-relaxation factors, or other solution parameters that you have set, as described in detail in teh following sections.

16.1.3.1.4. Reordering Species

If you find that the last species in the **Selected Species** list is not the most abundant species (as it should be), you will need to rearrange the species to obtain the proper order.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Remove the bulk species from the **Selected Species** list. It will now appear in the **Available Species** list.
2. Add the species back in again. It will automatically be placed at the end of the list.

16.1.3.1.5. The Naming and Ordering of Species

As discussed previosly, you should retain the most abundant species as the last one in the **Selected Species** list when you add or remove species. Additional considerations you should be aware of when adding and deleting species are presented here.

There are three characteristics of a species that identify it to the solver: name, chemical formula, and position in the list of species in the **Species** dialog box. Changing these characteristics will have the following effects:

- You can change the **Name** of a species (using the *Create/Edit Materials Dialog Box* (p. 1736), as described in *Renaming an Existing Material* (p. 388)) without any consequences.
- You should *never* change the given **Chemical Formula** of a specie.
- You will change the order of the species list if you add or remove any specie. When this occurs, all cell zone or boundary conditions, solver parameters, and solution data for species will be reset to the default values. (Solution data, cell zone or boundary conditions, and solver parameters for other flow variables will not be affected.) Thus, if you add or remove species you should take care to redefine species cell zone and boundary conditions and solution parameters for the newly defined problem. In addition, you should recognize that patched species concentrations or concentrations stored in any data file that was based on the original species ordering will be incompatible with the newly defined problem. You can use the data file as a starting guess, but you should be aware that the species concentrations in the data file may provide a poor initial guess for the newly defined model.

16.1.3.2. Defining Reactions

If your ANSYS FLUENT model involves chemical reactions, you can define the reactions in which the defined species participate. This will be necessary only if you are creating a mixture material from scratch, you have modified the species, or you want to redefine the reactions for some other reason.

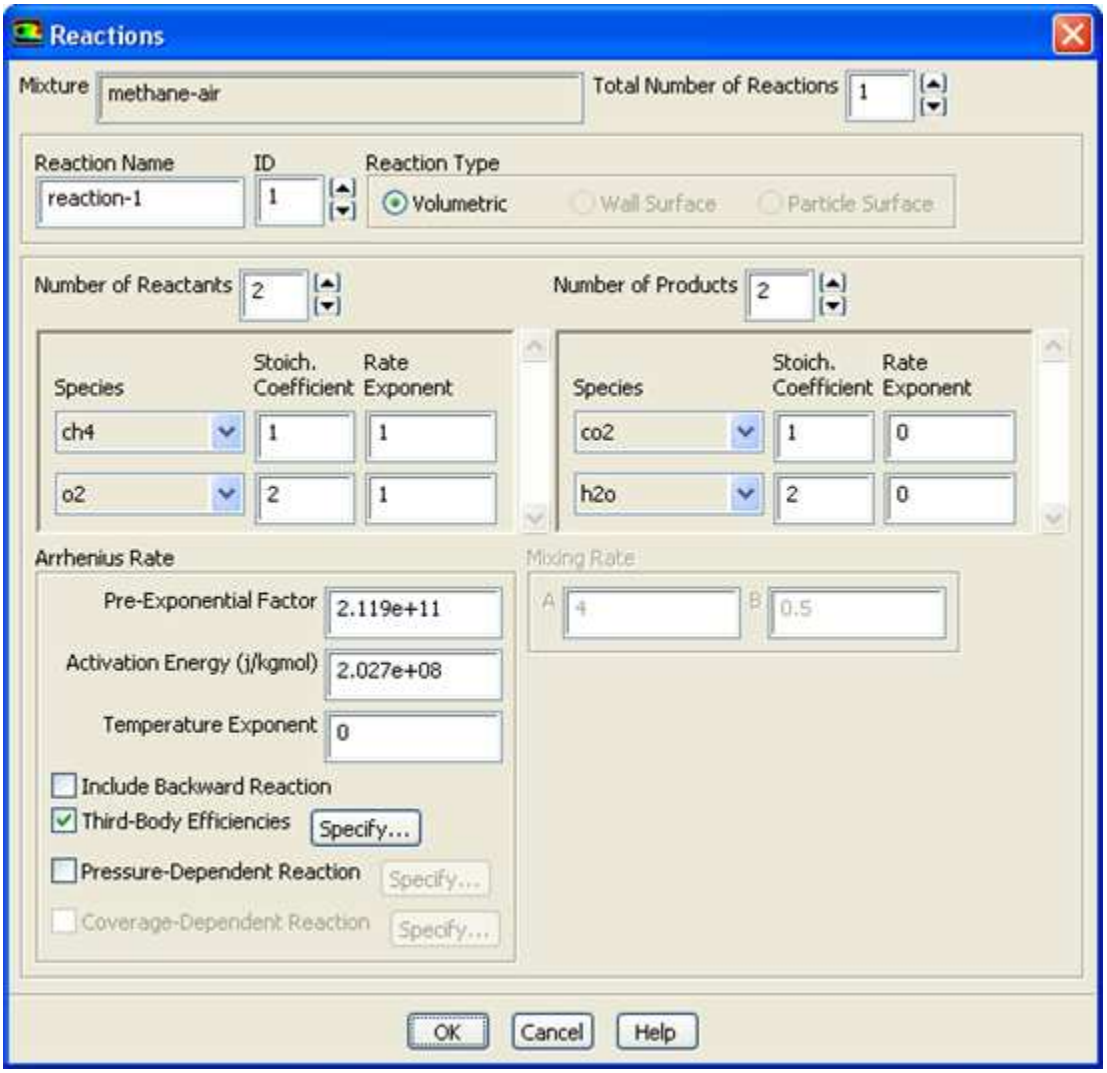
Depending on which turbulence-chemistry interaction model you selected in the **Species Model** dialog box (see *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801)), the appropriate reaction model will be displayed in the **Reaction** drop-down list in the **Edit Material** dialog box. If you are using the **Laminar Finite-Rate** or **Eddy-Dissipation Concept** model, the reaction model will be **finite-rate**; if you are using the **Eddy-Dissipation** model, the reaction model will be **eddy-dissipation**; if you are using the **Finite-Rate/Eddy-Dissipation** model, the reaction model will be **finite-rate/eddy-dissipation**.

16.1.3.2.1. Inputs for Reaction Definition

To define the reactions, click the **Edit...** button to the right of **Reaction**. The **Reactions** dialog box (*Figure 16.5* (p. 810)) will open.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.5 The Reactions Dialog Box



The steps for defining reactions are as follows:

1. Set the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions) in the **Total Number of Reactions** field. Use the arrows to change the value, or type in the value and press `Enter`.

   Note that if your model includes discrete-phase combusting particles, you should include the particulate surface reaction(s) (e.g., char burnout, multiple char oxidation) in the number of reactions *only* if you plan to use the multiple surface reactions model for surface combustion.

2. Specify the **Reaction Name** of the reaction you want to define.

3. Set the **ID** of the reaction you want to define. Again, if you type in the value be sure to press `Enter`.

4. If this is a fluid-phase reaction, keep the default selection of **Volumetric** as the **Reaction Type**. If this is a wall surface reaction (described in *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829)) or a particle surface reaction (described in *Particle Surface Reactions* (p. 835)), select **Wall Surface** or **Particle Surface** as the **Reaction Type**. See *User Inputs for Particle Surface Reactions* (p. 835) for further information about defining particle surface reactions.

5. Specify how many reactants and products are involved in the reaction by increasing the value of the **Number of Reactants** and the **Number of Products**. Select each reactant or product in the **Species**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drop-down list and then set its stoichiometric coefficient and rate exponent in the appropriate **Stoich. Coefficient** and **Rate Exponent** fields. The stoichiometric coefficient is the constant $v'_{i,r}$ or $v''_{i,r}$ in Equation 7–6 in the Theory Guide and the rate exponent is the exponent on the reactant or product concentration, $\eta'_{j,r}$ or $\eta''_{j,r}$ in Equation 7–8 in the Theory Guide.

There are two general classes of reactions that can be handled by the **Reactions** dialog box, so it is important that the parameters for each reaction are entered correctly. The classes of reactions are as follows:

- Global forward reaction (no reverse reaction): Product species generally do not affect the forward rate, so the rate exponent for all products ($\eta''_{j,r}$) should be 0. For reactant species, set the rate exponent ($\eta'_{j,r}$) to the desired value. If such a reaction is not an elementary reaction, the rate exponent will generally not be equal to the stoichiometric coefficient ($v'_{i,r}$) for that specie. An example of a global forward reaction is the combustion of methane:

$$CH_4 + 2O_2 \rightarrow CO_2 + 2H_2O \qquad \textbf{(16–1)}$$

where $v'_{CH_4} = 1, \eta'_{CH_4} = 0.2, v'_{O_2} = 2, \eta'_{O_2} = 1.3, v''_{CO_2} = 1, \eta''_{CO_2} = 0, v''_{H_2O} = 2$, and $\eta''_{H_2O} = 0$.

Figure 16.5 (p. 810) shows the coefficient inputs for the combustion of methane. See also the **methane-air** mixture material in the *FLUENT Database Materials Dialog Box* (p. 1744).

Note that, in certain cases, you may wish to model a reaction where product species affect the forward rate. For such cases, set the product rate exponent ($\eta''_{j,r}$) to the desired value. An example of such a reaction is the gas-shift reaction (see the **carbon-monoxide-air** mixture material in the *FLUENT Database Materials Dialog Box* (p. 1744)), in which the presence of water has an effect on the reaction rate:

$$CO + \frac{1}{2}O_2 + H_2O \rightarrow CO_2 + H_2O$$

In the gas-shift reaction, the rate expression may be defined as:

$$k\,[CO]\,[O_2]^{1/4}\,[H_2O]^{1/2} \qquad \textbf{(16–2)}$$

where $v'_{CO} = 1, \eta'_{CO} = 1, v'_{O_2} = 0.5, \eta'_{O_2} = 0.25, v''_{CO_2} = 1, \eta''_{CO_2} = 0, v''_{H_2O} = 0$, and $\eta''_{H_2O} = 0.5$.

- Reversible reaction: An elementary chemical reaction that assumes the rate exponent for each species is equivalent to the stoichiometric coefficient for that specie. An example of an elementary reaction is the oxidation of SO_2 to SO_3:

$$SO_2 + \frac{1}{2}O_2 \rightleftharpoons SO_3$$

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $v'_{SO_2}=1, \eta'_{SO_2}=1, v'_{O_2}=0.5, \eta'_{O_2}=0.5, v''_{SO_3}=1$, and $\eta''_{SO_3}=1$.

See step 6 below for information about how to enable reversible reactions.

6. If you are using the laminar finite-rate, finite-rate/eddy-dissipation, Eddy-Dissipation Concept or PDF Transport model for the turbulence-chemistry interaction, enter the following parameters for the Arrhenius rate in the **Arrhenius Rate** group box:

Pre-Exponential Factor

(the constant A_r in Equation 7–10 in the Theory Guide). The units of A_r must be specified such that the units of the molar reaction rate, $\hat{R}_{i,r}$ in Equation 7–5 in the Theory Guide, are moles/volume-time (e.g., kmol/m^3-s) and the units of the volumetric reaction rate, R_i in Equation 7–5 in the Theory Guide, are mass/volume-time (e.g., kg/m^3-s).

Important

It is important to note that if you have selected the British units system, the Arrhenius factor should still be input in SI units. This is because ANSYS FLUENT applies no conversion factor to your input of A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your inputs for $v'_{i,r}$, β_r, etc.

Activation Energy

(the constant E_r in the forward rate constant expression, Equation 7–10 in the Theory Guide).

Temperature Exponent

(the value for the constant β_r in Equation 7–10 in the Theory Guide).

Third-Body Efficiencies

(the values for $\gamma_{j,r}$ in Equation 7–9 in the Theory Guide). If you have accurate data for the efficiencies and want to include this effect on the reaction rate (i.e., include Γ in Equation 7–8 in the Theory Guide), enable the **Third Body Efficiencies** option and click the **Specify...** button to open the *Third-Body Efficiencies Dialog Box* (p. 1768) (*Figure 16.6* (p. 813)).

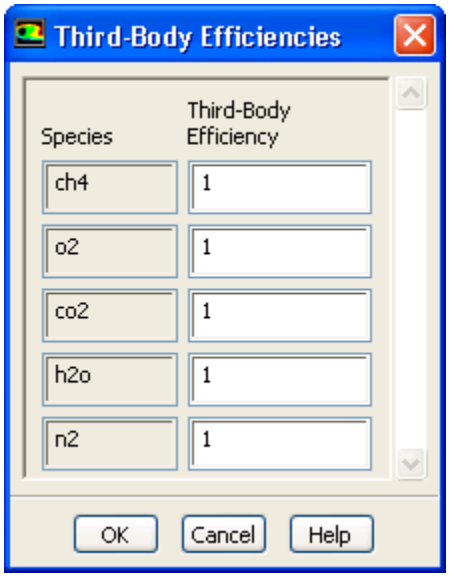
For each **Species** in the dialog box, specify the **Third-Body Efficiency**.

Important

It is not necessary to include the third-body efficiencies. You should not enable the **Third-Body Efficiencies** option unless you have accurate data for these parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.6 The Third-Body Efficiencies Dialog Box

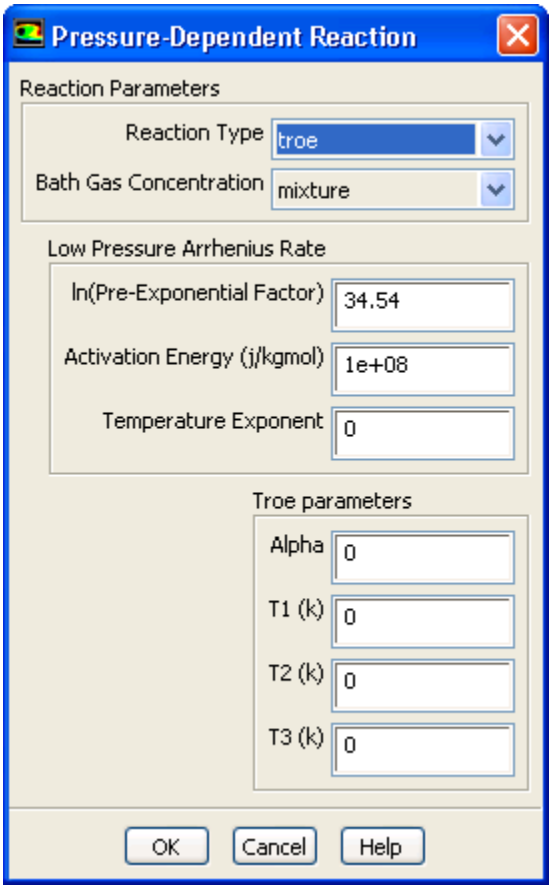


Pressure-Dependent Reaction

(if relevant) If you are using the laminar finite-rate or Eddy-Dissipation Concept model for turbulence-chemistry interaction, or have enabled the composition PDF transport model (see *Modeling a Composition PDF Transport Problem* (p. 911)), and the reaction is a pressure fall-off reaction (see Pressure-Dependent Reactions in the Theory Guide), enable the **Pressure-Dependent Reaction** option for the **Arrhenius Rate** and click the **Specify...** button to open the **Pressure-Dependent Reaction** dialog box (*Figure 16.7* (p. 814)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.7 The Pressure-Dependent Reaction Dialog Box



Under **Reaction Parameters**, select the appropriate **Reaction Type** (**lindemann**, **troe**, or **sri**). See Pressure-Dependent Reactions in the Theory Guide for details about the three methods. Next, specify if the **Bath Gas Concentration** ($[M]$ in Equation 7–18 in the Theory Guide) is to be defined as the concentration of the **mixture**, or as the concentration of one of the mixture's constituent species, by selecting the appropriate item in the drop-down list.

The parameters you specified under **Arrhenius Rate** in the **Reactions** dialog box represent the high-pressure Arrhenius parameters. You can, however, specify values for the following parameters under **Low Pressure Arrhenius Rate**:

ln(Pre-Exponential Factor)

(A_{low} in Equation 7–16 in the Theory Guide) The pre-exponential factor A_{low} is often an extremely large number, so you will input the natural logarithm of this term.

Activation Energy

(E_{low} in Equation 7–16 in the Theory Guide)

Temperature Exponent

(β_{low} in Equation 7–16 in the Theory Guide)

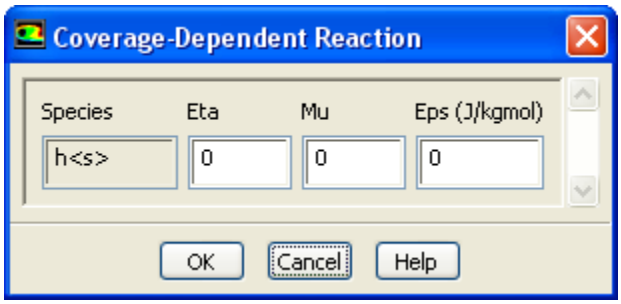
If you selected **troe** for the **Reaction Type**, you can specify values for **Alpha**, **T1**, **T2**, and **T3** (α, T_1, T_2, and T_3 in Equation 7–21 in the Theory Guide) under **Troe parameters**. If you selected **sri** for the **Reaction Type**, you can specify values for **a**, **b**, **c**, **d**, and **e** (a, b, c, d, and e in Equation 7–22 in the Theory Guide) under **SRI parameters**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coverage Dependent Reaction

If you are modeling **Wall Surface** reactions with site-balancing and you have reaction rates that depend on site coverages, you can enable the **Coverage Dependent Reaction** option. Click **Specify...** to open the **Coverage Dependent Reaction** dialog box (*Figure 16.8* (p. 815)) and input the coverage parameters.

Figure 16.8 The Coverage Dependent Reaction Dialog Box



In the **Coverage Dependent Reaction** dialog box, all the site species of the reaction will be present with a default value of 0 for all the parameters, corresponding to no surface coverage modification. Enter the relevant values of the parameters μ, ε, and η (as defined in Equation 7–45 in the Theory Guide) for all the species for which the reaction has coverage dependence.

7. If you are using the laminar finite-rate, Eddy-Dissipation Concept or PDF Transport model for turbulence-chemistry interaction, and the reaction is reversible, enable the **Include Backward Reaction** option for the **Arrhenius Rate**. When this option is enabled, you will not be able to edit the **Rate Exponent** for the product species, which instead will be set to be equivalent to the corresponding product **Stoich. Coefficient**. If you do not wish to use ANSYS FLUENT's default values, or if you are defining your own reaction, you will also need to specify the standard-state enthalpy and standard-state entropy, to be used in the calculation of the backward reaction rate constant (Equation 7–11 in the Theory Guide). Note that the reversible reaction option is not available for either the eddy-dissipation or the finite-rate/eddy-dissipation turbulence-chemistry interaction model.

8. If you are using the eddy-dissipation or finite-rate/eddy-dissipation model for turbulence-chemistry interaction, you can enter values for **A** and **B** under the **Mixing Rate** heading. These values should not be changed unless you have reliable data. In most cases you will use the default values.

   A is the constant A in the turbulent mixing rate (Equation 7–24 and Equation 7–25 in the Theory Guide) when it is applied to a species that appears as a reactant in this reaction. The default setting of 4.0 is based on the empirically derived values given by Magnussen et al. [*49*] (p. 2211).

   B is the constant B in the turbulent mixing rate (Equation 7–25 in the Theory Guide) when it is applied to a species that appears as a product in this reaction. The default setting of 0.5 is based on the empirically derived values given by Magnussen et al. [*49*] (p. 2211).

9. Repeat steps 2–8 for each reaction you need to define. After defining all reactions, click **OK**.

16.1.3.2.2. Defining Species and Reactions for Fuel Mixtures

Quite often, combustion systems will include fuel that is not easily described as a pure species (such as CH_4 or C_2H_6). Complex hydrocarbons, including fuel oil or even wood chips, may be difficult to define in terms of such pure species. However, if you have available the heating value and the ultimate analysis (elemental composition) of the fuel, you can define an equivalent fuel species and an equivalent heat of formation for this fuel. Consider, for example, a fuel known to contain 50% C, 6% H, and 44% O by weight. Dividing by

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

atomic weights, you can arrive at a "fuel" species with the molecular formula $C_{4.17}H_6O_{2.75}$. You can start from a similar, existing species or create a species from scratch, and assign it a molecular weight of 100.04 kg/kmol (4.17 × 12 + 6 × 1 + 2.75 × 16). The chemical reaction would be considered to be

$$C_{4.17}H_6O_{2.75}+4.295O_2 \rightarrow 4.17CO_2+3H_2O \tag{16–3}$$

You will need to set the appropriate stoichiometric coefficients for this reaction.

The heat of formation (or standard-state enthalpy) for the fuel species can be calculated from the known heating value ΔH since

$$\Delta H = \sum_{i=1}^{N} h_i^0 \left(v_{i,r}'' - v_{i,r}' \right) \tag{16–4}$$

where h_i^0 is the standard-state enthalpy on a molar basis. Note the sign convention in *Equation 16–4* (p. 816) : ΔH is negative when the reaction is exothermic.

16.1.3.3. Defining Zone-Based Reaction Mechanisms

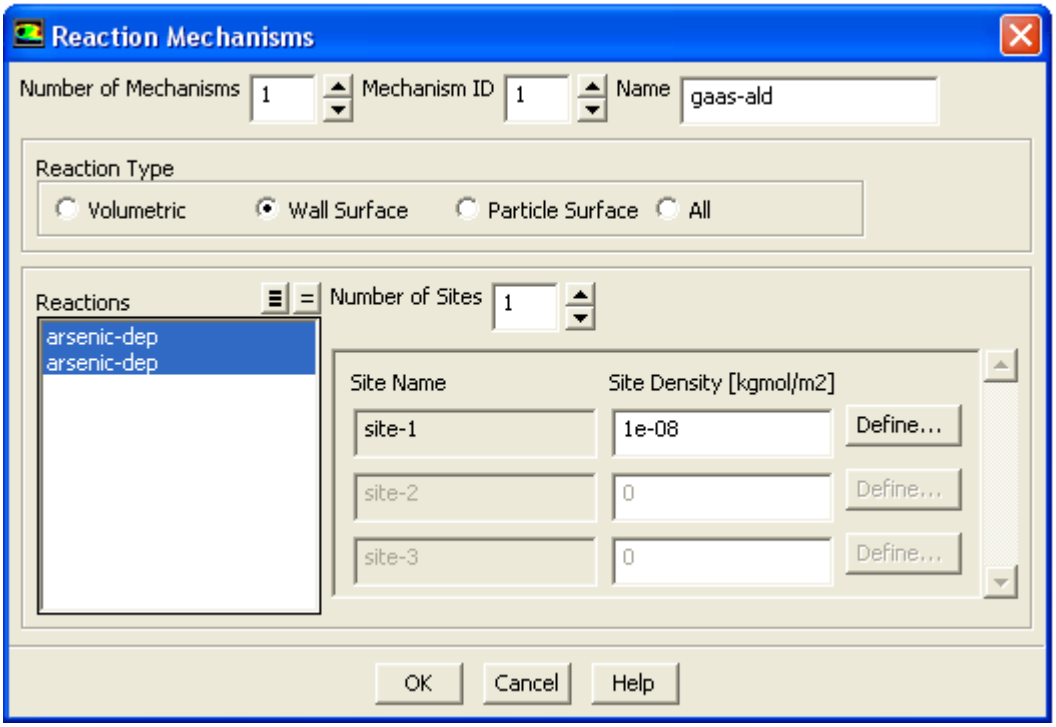
If your ANSYS FLUENT model involves reactions that are confined to a specific area of the domain, you can define "reaction mechanisms" to enable different reactions selectively in different geometrical zones. You can create reaction mechanisms by selecting reactions from those defined in the **Reactions** dialog box and grouping them. You can then assign a particular mechanism to a particular zone.

16.1.3.3.1. Inputs for Reaction Mechanism Definition

To define a reaction mechanism, click the **Edit...** button to the right of **Mechanism**. The *Reaction Mechanisms Dialog Box* (p. 1771) (*Figure 16.9* (p. 817)) will open.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.9 The Reaction Mechanisms Dialog Box



The steps for defining a reaction mechanism are as follows:

1. Set the total number of mechanisms in the **Number of Mechanisms** field. Use the arrows to change the value, or type the value and press `Enter`.
2. Set the **Mechanism ID** of the mechanism you want to define. Again, if you type in the value, be sure to press `Enter`.
3. Specify the **Name** of the mechanism.
4. Select the type of reaction to add to the mechanism under **Reaction Type**. If you select **Volumetric**, the **Reactions** list will display all available fluid-phase reactions. If you select **Wall Surface** or **Particle Surface**, the **Reactions** list will display all available wall surface reactions (described in *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829)) or particle surface reactions (described in *Particle Surface Reactions* (p. 835)). If you select **All**, the **Reactions** list will display all available reactions. This option is meant for backward compatibility with ANSYS FLUENT 6.0 or earlier cases.
5. Select the reactions to be included in the mechanism.
   - For **Volumetric** or **Particle Surface** reactions, select available reactions for the mechanism in the **Reactions** list.
   - For **Wall Surface** reactions, use the following procedure:
     a. Select available wall surface reactions for the mechanism in the **Reactions** list.
     b. If any site species appear in the selected reaction(s), set the number of sites in the **Number of Sites** field. Use the arrows to change the value, or type the value and press `Enter`. See Reaction-Diffusion Balance for Surface Chemistry in the Theory Guide for details about site species in wall surface reactions.
     c. If you specify a **Number of Sites** that is greater than zero, specify the properties of the site.

     Site Name
     (optional)

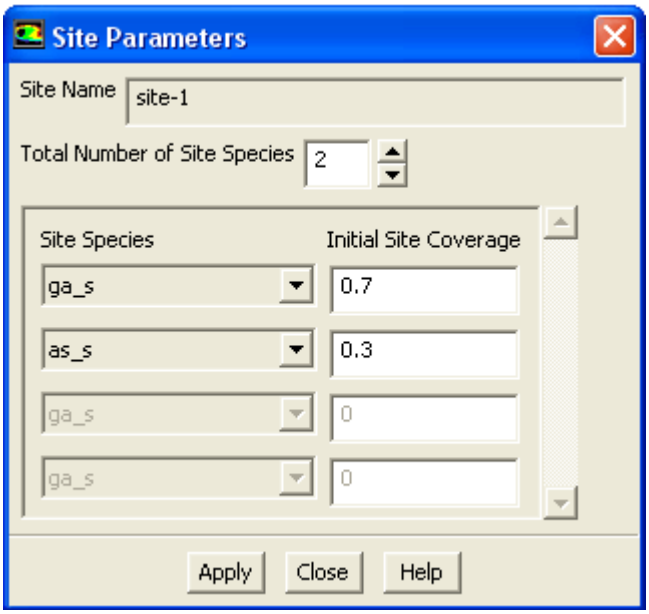
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Site Density

(in kmol/m^2) This value is typically in the range of 10^{-8} to 10^{-6}.

Click the **Define...** button. This will open the *Site Parameters Dialog Box* (p. 1772) (*Figure 16.10* (p. 818)), where you will define the parameters of the site specie.

Figure 16.10 The Site Parameters Dialog Box



Site Name

is the optional name of the site that was specified in the **Reaction Mechanisms** dialog box.

Total Number of Site Species

is the number of adsorbed species that are to be modeled at the site. (Use the arrows to change the value, or type the value and press `Enter`.)

Under **Site Species**, select the appropriate species from the drop-down list(s) and specify the fractional **Initial Site Coverage** for each specie. For steady-state calculations, it is recommended (though not strictly required) that the initial values of **Initial Site Coverage** sum to unity. For transient calculations, it is required that these values sum to unity.

Click **Apply** in the **Site Parameters** dialog box to store the new values.

6. Repeat steps 2–5 for each reaction mechanism you need to define. When you are finished defining all reaction mechanisms, click **OK**.

16.1.3.4. Defining Physical Properties for the Mixture

When your ANSYS FLUENT model includes chemical species, the following physical properties must be defined, either by you or by the database, for the mixture material:

- density, which you can define using the gas law or as a volume-weighted function of composition
- viscosity, which you can define as a function of composition
- thermal conductivity and specific heat (in problems involving solution of the energy equation), which you can define as functions of composition

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- mass diffusion coefficients and Schmidt number, which govern the mass diffusion fluxes (Equation 7–2 and Equation 7–3 in the Theory Guide)

Detailed descriptions of these property inputs are provided in *Physical Properties* (p. 385).

Important

Remember to click the **Change/Create** button when you are done setting the properties of the mixture material. The properties that appear for each of the constituent species will depend on your settings for the properties of the mixture material. If, for example, you specify a composition-dependent viscosity for the mixture, you will need to define viscosity for each specie.

16.1.3.5. Defining Physical Properties for the Species in the Mixture

For each of the fluid materials in the mixture, you (or the database) must define the following physical properties:

- molecular weight, which is used in the gas law and/or in the calculation of reaction rates and mole-fraction inputs or outputs
- standard-state (formation) enthalpy and reference temperature (in problems involving solution of the energy equation)
- viscosity, if you defined the viscosity of the mixture material as a function of composition
- thermal conductivity and specific heat (in problems involving solution of the energy equation), if you defined these properties of the mixture material as functions of composition
- standard-state entropy, if you are modeling reversible reactions
- thermal and momentum accommodation coefficients, if you have enabled the low-pressure boundary slip model.

Detailed descriptions of these property inputs are provided in *Physical Properties* (p. 385).

Important

Global reaction mechanisms with one or two steps inevitably neglect the intermediate species. In high-temperature flames, neglecting these dissociated species may cause the temperature to be overpredicted. A more realistic temperature field can be obtained by increasing the specific heat capacity for each specie. Rose and Cooper [*70*] (p. 2212) have created a set of specific heat polynomials as a function of temperature.

The specific heat capacity for each species is calculated as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$c_p(T) = \sum_{k=0}^{m} a_k T^k \tag{16–5}$$

The modified c_p polynomial coefficients (J/kg-K) from [64] (p. 2212) are provided in *Table 16.1: Modified Specific Heat Capacity (Cp) Polynomial Coefficients (J/kg-K)* (p. 820) and *Table 16.2: Modified Specific Heat Capacity (Cp) Polynomial Coefficients* (p. 820).

Table 16.1 Modified Specific Heat Capacity (Cp) Polynomial Coefficients (J/kg-K)

	N_2	CH_4	CO	H_2
a_0	1.02705e+03	2.00500e+03	1.04669e+03	1.4147e+04
a_1	2.16182e −02	−6.81428e −01	−1.56841e −01	1.7372e −01
a_2	1.48638e −04	7.08589e −03	5.39904e −04	6.9e −04
a_3	−4.48421e −08	−4.71368e −06	−3.01061e −07	—–
a_4	—–	8.51317e −10	5.05048e −11	—–

Table 16.2 Modified Specific Heat Capacity (Cp) Polynomial Coefficients

	CO_2	H_2O	O_2
a_0	5.35446e+02	1.93780e+03	8.76317e+02
a_1	1.27867e+00	−1.18077e+00	1.22828e −01
a_2	−5.46776e −04	3.64357e −03	5.58304e −04
a_3	−2.38224e −07	−2.86327e −06	−1.20247e −06
a_4	1.89204e −10	7.59578e −10	1.14741e −09
a_5	—–	—–	−5.12377e −13
a_6	—–	—–	8.56597e −17

16.1.4. Setting up Coal Simulations with the Coal Calculator Dialog Box

The **Coal Calculator** dialog box automates calculation and setting of the relevant input parameters for the Species, Discrete-Phase (DPM) and Pollutant models associated with coal combustion. It is available in the **Species** dialog box for the **Species Transport** model when the **Eddy-Dissipation** or **Finite-Rate/Eddy-Dissipation** turbulence-chemistry option is selected.

The inputs to the **Coal Calculator** dialog box are:

1. Coal **Proximate Analysis**, which is the mass fraction of **Volatile, Fixed Carbon, Ash** and **Moisture** in the coal. ANSYS FLUENT will normalize the mass fractions so that they sum to unity.
2. Coal **Ultimate Analysis**, which is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal. ANSYS FLUENT will normalize the mass fractions so that they sum to unity.
3. A choice of **One-step** or **Two-step** chemical mechanism. The one-step mechanism is,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$vol + v_{O_2} O_2 \rightarrow v_{CO_2} CO_2 + v_{H_2O} H_2O + v_{SO_2} SO_2 + v_{N_2} N_2 \quad \textbf{(16–6)}$$

The two-step mechanism involves oxidation of volatiles to CO in the first reaction and oxidation of CO to CO_2 in the second reaction:

$$\begin{aligned} vol + v_{O_2} O_2 &\rightarrow v_{CO} CO + v_{CO_2} CO_2 + v_{H_2O} H_2O + v_{SO_2} SO_2 + v_{N_2} N_2 \\ CO + 0.5 O_2 &\rightarrow CO_2 \end{aligned} \quad \textbf{(16–7)}$$

The stoichiometric co-efficients in *Equation 16–6* (p. 821) and *Equation 16–7* (p. 821) are calculated from the ultimate and proximate analyses.

4. An option to **Include SO2**. When this is enabled, an input for the atomic mass fraction of sulphur, **S**, appears in the ultimate analysis frame.
5. **Wet Combustion**, which will enable the DPM Wet Combustion option by default in all injections created after the **OK** button is clicked in the Coal Calculator dialog box.
6. The **Coal Particle Material Name**. A DPM combusting-particle material will be created with this name. The default name is *coal-particle*.
7. The **Coal As-Received HCV**, where HCV denotes the Higher Calorific Value.
8. **Volatile Molecular Weight** is the molecular weight of pure volatiles.
9. The **CO/CO2 Split in Reaction 1 Products** can be used to specify the molar fraction of CO to CO_2 in the first reaction of *Equation 16–7* (p. 821). The default value of 1 implies that all carbon is reacted to CO, with no CO_2 produced.
10. The **High Temperature Volatile Yield**. Enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction. To model this, the actual volatile fraction used is calculated as that specified in the **Proximate Analysis** input multiplied by the **High Temperature Volatile Yield**. The actual fixed carbon fraction is then calculated as one minus the sum of the actual volatile, ash and moisture fractions.
11. **Fraction of N in Char (DAF)**. This input is used in calculating the split of atomic nitrogen for the Fuel NOx model.
12. **Coal Dry Density** is used to calculate the volume fraction of liquid-water for the **Wet Combustion** option in the **Injections** dialog box.

When **OK** is clicked, ANSYS FLUENT makes the following changes:

1. A Mixture material is created, named *coal-volatiles-air*, with a one or two step reaction mechanism as specified in the **Mechanism** option. If the Fluid material species (O_2, CO, CO_2 etc.) do not exist, they are created. A Fluid material called *coal-volatiles*, is also created with a standard state enthalpy calculated from the ultimate and proximate analyses, as-received HCV and volatile molecular weight.
2. A combusting-particle material is created with **Volatile Component Fraction** and **Combustible Fraction** calculated from the ultimate and proximate analyses. The discrete phase model (DPM) is enabled.
3. For the fuel NOx model, the default fuel species is set to *vol*, the char N conversion is set to *NO*, and the fuel NOx **Volatile** and **Char** mass fractions are set according to the ultimate and proximate compositions. Note that even though some of the default fuel NOx parameters are changed, the fuel NOx model itself is not enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If **Wet Combustion** is selected, all subsequent injections that are created will have wet combustion enabled. The evaporation material will be set to *water-liquid*, and the volume fraction of water will be calculated from the **Moisture** mass fraction specified in the proximate analysis, and the **Coal Dry Density**. The **Density** for the combusting-particle in the **Create/Edit Materials** dialog box will also be set to **Coal Dry Density**.

16.1.5. Defining Cell Zone and Boundary Conditions for Species

You will need to specify the inlet mass fraction for all species in your simulation. In addition, for pressure outlets you will set species mass fractions to be used in case of backflow. At walls, ANSYS FLUENT will apply a zero-gradient (zero-flux) boundary condition for all species by default, although you can change each species boundary condition to a specified value. If you have surface reactions defined (see *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829)), you can choose to enable wall-surface reactions and select the chemical mechanism. For fluid zones, you also have the option of specifying a reaction mechanism. Input of cell zone and boundary conditions is described in *Cell Zone and Boundary Conditions* (p. 205).

Important

- Non-reflecting boundary conditions (NRBCs) are not compatible with species transport models. They are mainly used to solve ideal-gas single species flow. For information about NRBCs, see *Non-Reflecting Boundary Conditions* (p. 339).
- Note that you will explicitly set mass fractions only for the first $N-1$ species. The solver will compute the mass fraction of the last species by subtracting the total of the specified mass fractions from 1. If you want to explicitly specify the mass fraction of the last specie, you must reorder the species in the list (in the *Create/Edit Materials Dialog Box* (p. 1736)), as described in *Defining Properties for the Mixture and Its Constituent Species* (p. 805).

16.1.5.1. Diffusion at Inlets with the Pressure-Based Solver

For the pressure-based solver in ANSYS FLUENT, the net transport of species at inlets consists of both convection and diffusion components. The convection component is fixed by the specified inlet species mass or mole fraction, whereas the diffusion component depends on the gradient of the computed species concentration field (which is not known a priori). At very small convective inlet velocities, for example when modeling perforated combustion liners with an inlet, substantial mass can be gained or lost through the inlet due to diffusion. For this reason, inlet diffusion is disabled by default, but can be enabled with the **Inlet Diffusion** option in the **Species Model** dialog box.

Models → **Species** → **Edit...**

16.1.6. Defining Other Sources of Chemical Species

You can define a source or sink of a chemical species within the computational domain by defining a source term in the **Fluid** dialog box. You may choose this approach when species sources exist in your problem but you do not want to model them through the mechanism of chemical reactions. *Defining Mass, Momentum, Energy, and Other Sources* (p. 244) describes the procedures you would follow to define species sources in your ANSYS FLUENT model. If the source is not a constant, you can use a user-defined function. See the UDF Manual for details about user-defined functions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.1.7. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry

While many simulations involving chemical species may require no special procedures during the solution process, you may find that one or more of the solution techniques noted in this section helps to accelerate the convergence or improve the stability of more complex simulations. The techniques outlined below may be of particular importance if your problem involves many species and/or chemical reactions, especially when modeling combusting flows.

16.1.7.1. Stability and Convergence in Reacting Flows

Obtaining a converged solution in a reacting flow can be difficult for a number of reasons. First, the impact of the chemical reaction on the basic flow pattern may be strong, leading to a model in which there is strong coupling between the mass/momentum balances and the species transport equations. This is especially true in combustion , where the reactions lead to a large heat release and subsequent density changes and large accelerations in the flow. All reacting systems have some degree of coupling, however, when the flow properties depend on the species concentrations. These coupling issues are best addressed by the use of a two-step solution process, as described below, and by the use of under-relaxation as described in *Setting Under-Relaxation Factors* (p. 1217).

A second convergence issue in reacting flows involves the magnitude of the reaction source term. When the ANSYS FLUENT model involves very rapid reaction rates (reaction time scales are much faster than convection and diffusion time scales), the solution of the species transport equations becomes numerically difficult. Such systems are termed "stiff" systems. Stiff systems with laminar chemistry can be solved using either the pressure-based solver with the **Stiff Chemistry Solver** option enabled, or the density-based solver (see *Solution of Stiff Laminar Chemistry Systems* (p. 824)). The laminar chemistry model may also be used for turbulent flames, where turbulence-chemistry interactions are neglected. However, for such flames, the Eddy-Dissipation Concept or PDF Transport models, which account for turbulence-chemistry interactions, may be a better choice.

16.1.7.2. Two-Step Solution Procedure (Cold Flow Simulation)

Solving a reacting flow as a two-step process can be a practical method for reaching a stable converged solution to your ANSYS FLUENT problem. In this process, you begin by solving the flow, energy, and species equations with reactions disabled (the "cold-flow", or unreacting flow). When the basic flow pattern has thus been established, you can re-enable the reactions and continue the calculation. The cold-flow solution provides a good starting solution for the calculation of the combusting system. This two-step approach to combustion modeling can be accomplished using the following procedure:

1. Set up the problem including all species and reactions of interest.
2. Temporarily disable reaction calculations by turning off **Volumetric** in the *Species Model Dialog Box* (p. 1681).

   Models → **Species** → **Edit...**
3. Disable calculation of the product species in the *Equations Dialog Box* (p. 1904).

   Solution Controls → **Equations...**
4. Calculate an initial (cold-flow) solution. (Note that it is generally not productive to obtain a fully converged cold-flow solution unless the non-reacting solution is also of interest to you.)
5. Enable the reaction calculations by turning on **Volumetric** again in the *Species Model Dialog Box* (p. 1681).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Enable all equations in the Equations dialog box. If you are using the laminar finite-rate, finite-rate/eddy-dissipation, Eddy-Dissipation Concept or PDF Transport model for turbulence-chemistry interaction, you may need to patch an ignition source (as described in *Ignition in Combustion Simulations* (p. 824)).

16.1.7.3. Density Under-Relaxation

One of the main reasons a combustion calculation can have difficulty converging is that large changes in temperature cause large changes in density, which can, in turn, cause instabilities in the flow solution. When you use the pressure-based solver, ANSYS FLUENT allows you to under-relax the change in density to alleviate this difficulty. The default value for density under-relaxation is 1, but if you encounter convergence trouble you may wish to reduce this to a value between 0.5 and 1 (in the **Solution Controls** task page).

16.1.7.4. Ignition in Combustion Simulations

If you introduce fuel to an oxidant, spontaneous ignition does not occur unless the temperature of the mixture exceeds the activation energy threshold required to maintain combustion. This physical issue manifests itself in an ANSYS FLUENT simulation as well. If you are using the laminar finite-rate, finite-rate/eddy-dissipation, Eddy-Dissipation Concept or PDF Transport model for turbulence-chemistry interaction, you have to supply an ignition source to initiate combustion. This ignition source may be a heated surface or inlet mass flow that heats the gas mixture above the required ignition temperature. Often, however, it is the equivalent of a spark: an initial solution state that causes combustion to proceed. You can supply this initial spark by patching a hot temperature into a region of the ANSYS FLUENT model that contains a sufficient fuel/air mixture for ignition to occur.

Solution Initialization → **Patch...**

Depending on the model, you may need to patch both the temperature and the fuel/ oxidant/product concentrations to produce ignition in your model. The initial patch has no impact on the final steady-state solution—no more than the location of a match determines the final flow pattern of the torch that it lights. See *Patching Values in Selected Cells* (p. 1242) for details about patching initial values.

16.1.7.5. Solution of Stiff Laminar Chemistry Systems

When modeling stiff laminar flames with the laminar finite-rate model, you can either use the pressure-based solver with the **Stiff Chemistry Solver** option enabled as seen in the *Species Model Dialog Box* (p. 1681) (*Figure 16.1* (p. 802)), or the density-based solver.

When using the pressure-based solver for unsteady simulations, the **Stiff Chemistry Solver** option applies a fractional step algorithm. In the first fractional step, the chemistry in each cell is reacted at constant pressure for the flow time-step, using the ISAT integrator. In the second fractional step, the convection and diffusion terms are treated just as in a non-reacting simulation.

For steady simulations using the pressure-based solver, the **Stiff Chemistry Solver** option approximates the reaction rate R_i in the species transport equation (see Equation 7–5 in the Theory Guide) as,

$$R_i^* = \frac{1}{\tau}\int_0^\tau R_i dt \qquad \textbf{(16–8)}$$

where τ is an appropriate time-step. Note that as τ tends to zero the approximation becomes exact but the stiff numerics will cause the pressure-based solver to diverge. On the other hand, as τ tends to infinity, the approximated reaction rate R_i^* tends to zero and, while the numerical stiffness is alleviated, there is no re-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

action. In ANSYS FLUENT, the default value for τ is set to one-tenth of the minimum convective or diffusive time-scale in the cell. This value was found to be sufficiently accurate and robust, although it can be modified using the `solve/set/stiff-chemistry` text command. ISAT is employed to integrate the stiff chemistry in *Equation 16–8* (p. 824).

Details about the ISAT algorithm may be found in Particle Reaction in the Theory Guide and *Using ISAT Efficiently* (p. 925). For efficient and accurate use of ISAT, a review of this section is highly recommended.

Choosing the density-based implicit solver can provide further solution stability by enabling the **Stiff Chemistry Solver** option. This option allows a larger stable Courant (CFL) number specification, although additional calculations are required to calculate the eigenvalues of the chemical Jacobian [*99*] (p. 2214). When enabling the stiff-chemistry solver, the following must be specified:

- **Temperature Positivity Rate Limit**: limits new temperature changes by this factor multiplied by the old temperature. Its default value is 0.2.
- **Temperature Time Step Reduction**: limits the local CFL number when the temperature is changing too rapidly. Its default value is 0.25.
- **Max. Chemical Time Step Ratio**: limits the local CFL number when the chemical time scales (eigenvalues of the chemical Jacobian) become too large to maintain a well-conditioned matrix. Its default value is 0.9.

If the density-based explicit solver is used, then the `stiff-chemistry` solver has to be enabled by using the text command:

`solve` → `set` → `stiff-chemistry`

You will be prompted to specify the following:

- Positivity Rate Limit (for temperature): limits new temperature changes by this factor multiplied by the old temperature. Its default value is 0.2.
- Temperature time-step reduction factor: limits the local CFL number when the temperature is changing too rapidly. Its default value is 0.25.
- Maximum allowable time-step/chemical-time-scale ratio: limits the local CFL number when the chemical time scales (eigenvalues of the chemical Jacobian) become too large to maintain a well-conditioned matrix. Its default value is 0.9.

The default values of these parameters are applicable in most cases.

16.1.7.6. Eddy-Dissipation Concept Model Solution Procedure

Due to the high computational expense of the Eddy-Dissipation Concept model, it is recommended that you use the following procedure to obtain a solution using the pressure-based solver:

1. Calculate an initial solution using the equilibrium Non-premixed or Partially-premixed model (see *Modeling Non-Premixed Combustion* (p. 839) and *Modeling Partially Premixed Combustion* (p. 905)).
2. Import a CHEMKIN format reaction mechanism (see *Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827)).
3. Enable the reaction calculations by turning on **Volumetric Reactions** in the **Species Model** dialog box and selecting **Eddy-Dissipation Concept** under **Turbulence-Chemistry Interaction**. Select the mechanism that you just imported as the **Mixture Material**.

   Models → **Species** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set the species boundary conditions.

   Boundary Conditions

5. Disable the flow and turbulence and solve for the species and temperature only.
6. Enable all equations and iterate to convergence. Note that the default numerical parameters for the solution of the Eddy-Dissipation Concept equations are set to provide maximum robustness with slowest convergence. The convergence rate can be increased by setting the `Acceleration Factor` in the **Species** dialog box or with the text command:

   `define` → `models` → `species` → `set-turb-chem-interaction`

   The `Acceleration Factor` can be set from 0 (slow but stable) to 1 (fast but least stable).

16.1.8. Postprocessing for Species Calculations

ANSYS FLUENT can report chemical species as mass fractions, mole fractions, and molar concentrations. You can also display laminar and effective mass diffusion coefficients. The following variables are available for postprocessing of species transport and reaction simulations:

- **Mass fraction of species-n**
- **Mole fraction of species-n**
- **Molar Concentration of species-n**
- **Lam Diff Coef of species-n**
- **Eff Diff Coef of species-n**
- **Thermal Diff Coef of species-n**
- **Enthalpy of species-n** (pressure-based solver calculations only)
- **species-n Source Term** (density-based solver calculations only)
- **Relative Humidity**
- **TFM Thickening Factor** (Thickened Flame Model only)
- **TFM Omega** (Thickened Flame Model only)
- **TFM Thickening Factor** (turbulent cases (LES/DES/SAS) with Thickened Flame Model only)
- **Laminar Flame Speed** (Thickened Flame Model only)
- **Laminar Flame Thickness** (Thickened Flame Model only)
- **Cell Time Scale** (Eddy-Dissipation Concept and Laminar finite-rate stiff-chemistry only)
- **Fine Scale Mass fraction of species-n** (Eddy-Dissipation Concept model only)
- **EDC Cell Volume Fraction** (Eddy-Dissipation Concept model only)
- **Fine Scale Temperature** (Eddy-Dissipation Concept model only)
- **Net Rate of species-n** (Eddy-Dissipation Concept and Laminar finite-rate stiff-chemistry only)
- **Kinetic Rate of Reaction-n**
- **Turbulent Rate of Reaction-n**
- **Liquid species mass fraction of species-n** (solidification and melting model only)
- **Heat of Reaction**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These variables are contained in the **Species...**, **Temperature...**, and **Reactions...** categories of the variable selection drop-down list that appears in postprocessing dialog boxes. See *Field Function Definitions* (p. 1529) for a complete list of flow variables, field functions, and their definitions. *Displaying Graphics* (p. 1379) and *Reporting Alphanumeric Data* (p. 1509) explain how to generate graphics displays and reports of data.

16.1.8.1. Averaged Species Concentrations

Averaged species concentrations at inlets and exits, and across selected planes (i.e., surfaces that you have created using the **Surface** menu items) within your model can be obtained using the *Surface Integrals Dialog Box* (p. 2031), as described in *Surface Integration* (p. 1519).

Reports → **Surface Integrals** → **Set Up...**

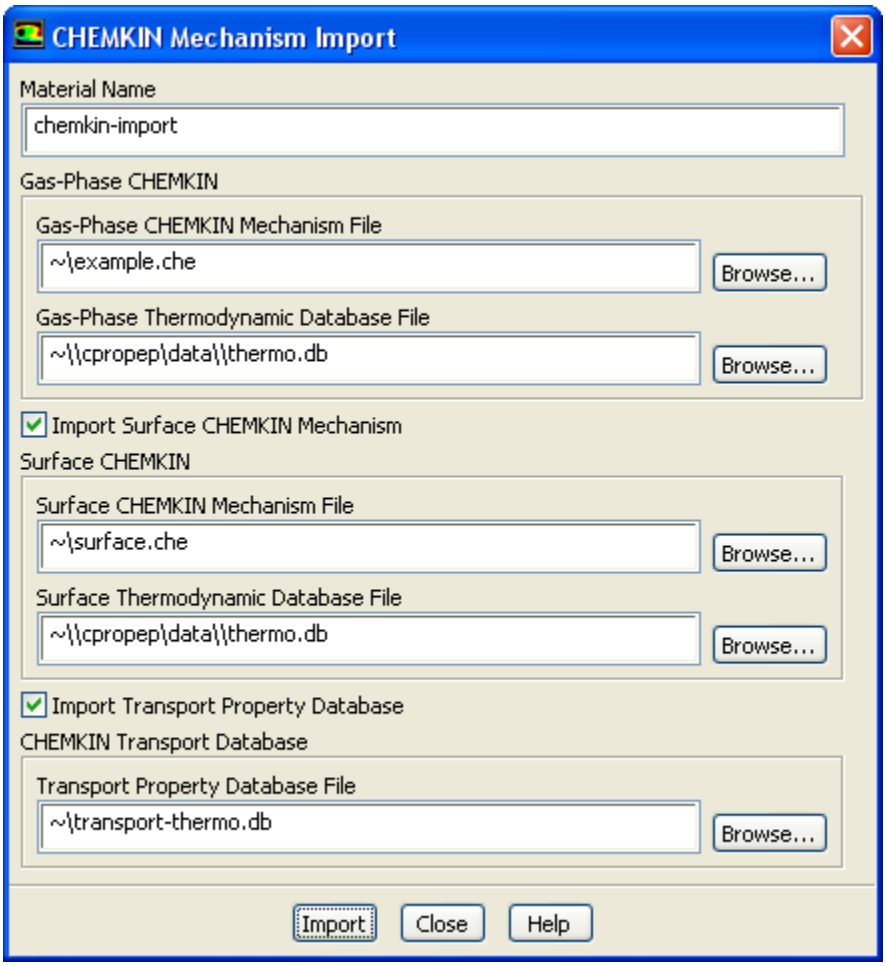
Select the **Molar Concentration of species-n** for the appropriate species in the **Field Variable** drop-down list.

16.1.9. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format

If you have a gas-phase chemical mechanism in CHEMKIN format, you can import the mechanism file into ANSYS FLUENT using the **CHEMKIN Mechanism Import** dialog box (*Figure 16.11* (p. 827)).

File → **Import** → **CHEMKIN Mechanism...**

Figure 16.11 The CHEMKIN Mechanism Import Dialog Box for Volumetric Kinetics



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

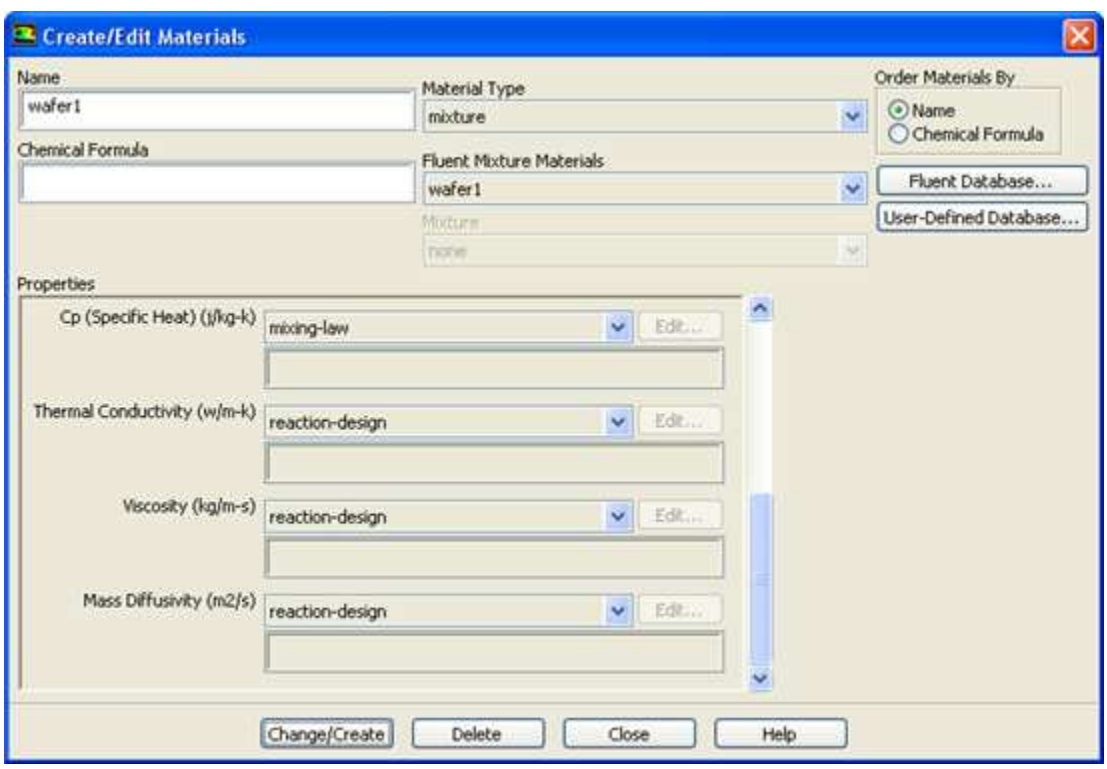
In the **CHEMKIN Mechanism Import** dialog box

1. Enter a name for the chemical mechanism under **Material Name**.
2. Enter the path to the CHEMKIN file (e.g., *path*/ `file.che`) under **Gas-Phase CHEMKIN Mechanism File**.
3. Specify the location of the **Gas-Phase Thermodynamic Database File** if the default thermodynamic database file, (`thermo.db`), does not contain all the gas-phase species in the CHEMKIN mechanism. The format for `thermo.db` is detailed in the CHEMKIN manual [*41*] (p. 2211).
4. (Optional). Import transport properties by enabling the **Import Transport Property Database** and entering the path to the CHEMKIN transport file. ANSYS FLUENT will enable mixture-averaged multi-component diffusion and set the Lennard-Jones kinetic theory parameters for all the species in the imported CHEMKIN mechanism. If you would like to use Stefan-Maxwell diffusion, enable the **Full Multicomponent Diffusion** in the **Species** dialog box.

   If you want to use KINetics transport properties, you must first enable the **CHEMKIN-CFD from Reaction Design** option in the **Species** dialog box (*Figure 16.1* (p. 802)). Once a file with KINetics transport properties is read, ANSYS FLUENT will create a material with the specified name, which will contain the data for the species and reactions, and add it to the list of available **Mixture Materials** in the *Create/Edit Materials Dialog Box* (p. 1736). For material properties such as **Specific Heat**, **Viscosity**, **Thermal Conductivity**, **Mass Diffusivity**, and **Thermal Diffusion**, listed in the **Create/Edit Materials** dialog box, the option **reaction-design** will appear in the drop-down list of each of the properties (*Figure 16.12* (p. 828)), allowing you to use material property values computed by KINetics.

   Note that when **Full Multicomponent Diffusion** is enabled in the **Species** dialog box, KINetics returns full multicomponent diffusivities. If **Full Multicomponent Diffusion** is disabled, KINetics returns mixture averaged mass diffusivities for each specie.

Figure 16.12 The Material Dialog Box When Importing CHEMKIN Transport Properties



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that since ANSYS FLUENT does not solve for the last specie, you should ensure that the last species in the CHEMKIN mechanism species list is the bulk specie. If not, edit the CHEMKIN mechanism file before importing it into ANSYS FLUENT, and move the bulk species (i.e. the species in your system with the largest total mass) to the end of the species list.

5. Click the **Import** button.

Important

Note that the CHEMKIN import facility does not provide full compatibility with all CHEMKIN rate formulations and that to access more complete functionality, you should consider the KINetics module option described in *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801).

For information on importing a surface kinetic mechanism in CHEMKIN format, see *Importing a Surface Kinetic Mechanism in CHEMKIN Format* (p. 832).

16.2. Wall Surface Reactions and Chemical Vapor Deposition

For gas-phase reactions, the reaction rate is defined on a volumetric basis and the rate of creation and destruction of chemical species becomes a source term in the species conservation equations. Heterogeneous surface reactions create sources (and sinks) of chemical species in the gas-phase as well, but also alter surface coverages for surface site reactions, and may deposit or etch species for bulk (solid) reactions.

For more information about the theoretical background of wall surface reactions and chemical vapor depositions, see Wall Surface Reactions and Chemical Vapor Deposition in the Theory Guide. Information about using wall surface reactions is presented in the following subsections:

16.2.1. Overview of Surface Species and Wall Surface Reactions
16.2.2. User Inputs for Wall Surface Reactions
16.2.3. Including Mass Transfer To Surfaces in Continuity
16.2.4. Wall Surface Mass Transfer Effects in the Energy Equation
16.2.5. Modeling the Heat Release Due to Wall Surface Reactions
16.2.6. Solution Procedures for Wall Surface Reactions
16.2.7. Postprocessing for Surface Reactions
16.2.8. Importing a Surface Kinetic Mechanism in CHEMKIN Format

16.2.1. Overview of Surface Species and Wall Surface Reactions

ANSYS FLUENT treats chemical species adsorbed and desorbed, as well as those deposited into or etched from the bulk solid surfaces as distinct from the same chemical species in the gas. Similarly, surface reactions are defined distinctly and treated differently than gas-phase reactions involving the same chemical species.

Surface reactions can be limited so that they occur on only some of the wall boundaries (while the other wall boundaries remain free of surface reaction). The surface reaction rate is defined and computed per unit surface area, in contrast to the fluid-phase reactions, which are based on unit volume.

16.2.2. User Inputs for Wall Surface Reactions

The basic steps for setting up a problem involving wall surface reactions are the same as those presented in *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799) for setting up a problem with only fluid-phase reactions, with a few additions:

1. In the *Species Model Dialog Box* (p. 1681):

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Models → **Species** → **Edit...**

a. Enable **Species Transport**, select **Volumetric** and **Wall Surface** under **Reactions**, and specify the **Mixture Material**. See *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801) for details about this procedure, and *Mixture Materials* (p. 800) for an explanation of the mixture material concept.

b. (optional) If you want to model the heat release due to wall surface reactions, enable the **Heat of Surface Reactions** option.

c. (optional) If you want to include the effect of surface mass transfer in the continuity equation, enable the **Mass Deposition Source** option.

d. To control the robustness and the convergence speed, enter a value between 0 and 1 for the **Aggressiveness Factor**. A value of 0 (the default) is the most robust, but results in the slowest convergence.

e. (optional) If you are using the pressure-based solver and you want to include species diffusion effects in the energy equation, enable the **Diffusion Energy Source** option. See *Wall Surface Mass Transfer Effects in the Energy Equation* (p. 831) for details.

f. (optional, but recommended for CVD) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option. See *Full Multicomponent Diffusion* (p. 440) for details.

2. Check and/or define the properties of the mixture. See *Defining Properties for the Mixture and Its Constituent Species* (p. 805).

Materials

Mixture properties include the following:

- species in the mixture
- reactions
- other physical properties (e.g., viscosity, specific heat)

Important

- You will find all species (including the solid/bulk and site species) in the list of **Fluent Fluid Materials**. For a deposited species such as Si, you will need both Si(g) and Si(s) in the materials list for the **fluid** material type.
- Note that the *final* gas-phase species named in the **Selected Species** list should be the carrier gas. This is because ANSYS FLUENT will not solve the transport equation for the final specie. Note also that any reordering, adding or deleting of species should be handled with caution, as described in *Reordering Species* (p. 808).

3. Check and/or set the properties of the individual species in the mixture. (See *Defining Properties for the Mixture and Its Constituent Species* (p. 805).) Note that if you are modeling the heat of surface reactions, you should be sure to check (or define) the formation enthalpy for each specie.

4. Set species boundary conditions.

Boundary Conditions

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition to the boundary conditions described in *Defining Cell Zone and Boundary Conditions for Species* (p. 822), you will first need to indicate whether or not surface reactions are in effect on each wall. If so, you will then need to assign a reaction mechanism to the wall. To enable the effect of surface reaction on a wall, enable the **Reaction** option in the **Species** section of the **Wall** dialog box.

Important

If you have enabled the global **Low-Pressure Boundary Slip** option in the *Viscous Model Dialog Box* (p. 1645), the **Shear Condition** for each wall will be reset to **No Slip** even though the slip model will be in effect. Note that the **Low-Pressure Boundary Slip** option is available only when the **Laminar** model is selected in the **Viscous Model** dialog box.

See *Inputs at Wall Boundaries* (p. 300) for details about boundary condition inputs for walls. See *User Inputs for Porous Media* (p. 225) for details about boundary condition inputs for porous media.

16.2.3. Including Mass Transfer To Surfaces in Continuity

In the surface reaction boundary condition described above, the effects of the wall normal velocity or bulk mass transfer to the wall are not included in the computation of species transport. The momentum of the net surface mass flux from the surface is also ignored because the momentum flux through the surface is usually small in comparison with the momentum of the flow in the cells adjacent to the surface. However, you can include the effect of surface mass transfer in the continuity equation by activating the **Mass Deposition Source** option in the *Species Model Dialog Box* (p. 1681).

16.2.4. Wall Surface Mass Transfer Effects in the Energy Equation

Species diffusion effects in the energy equation due to wall surface reactions are included in the normal species diffusion term described in Treatment of Species Transport in the Energy Equation in the Theory Guide.

If you are using the pressure-based solver, you can neglect this term by disabling the **Diffusion Energy Source** option in the *Species Model Dialog Box* (p. 1681). For the density-based solvers, this term is always included; you cannot disable it. Neglecting the species diffusion term implies that errors may be introduced to the prediction of temperature in problems involving mixing of species with significantly different heat capacities, especially for components with a Lewis number far from unity. While the effect of species diffusion should go to zero at Le = 1, you may see subtle effects due to differences in the numerical integration in the species and energy equations.

16.2.5. Modeling the Heat Release Due to Wall Surface Reactions

The heat release due to a wall surface reaction is, by default, ignored by ANSYS FLUENT. You can, however, choose to include the heat of surface reaction by activating the **Heat of Surface Reactions** option in the *Species Model Dialog Box* (p. 1681) and setting appropriate formation enthalpies in the *Edit Material Dialog Box* (p. 1782).

16.2.6. Solution Procedures for Wall Surface Reactions

As in all CFD simulations, your surface reaction modeling effort may be more successful if you start with a simple problem description, adding complexity as the solution proceeds. For wall surface reactions, you can follow the same guidelines presented for fluid-phase reactions in *Solution Procedures for Chemical Mixing and Finite-Rate Chemistry* (p. 823).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, if you are modeling the heat release due to surface reactions and you are having convergence trouble, you should try temporarily turning off the **Heat of Surface Reactions** and **Mass Deposition Source** options in the *Species Model Dialog Box* (p. 1681).

If you are modeling surface site species, good estimates of the **Initial Site Coverage** will aid convergence.

16.2.7. Postprocessing for Surface Reactions

In addition to the gas-phase variables listed in *Postprocessing for Species Calculations* (p. 826), for surface reactions you can display/report the surface coverage as well as the deposition rate of the solid species deposited on a surface. Select **Surface Coverage of species-n** or **Surface Deposition Rate of species-n** in the **Species...** category of the variable selection drop-down list.

Important

For surface reactions involving porous media, you can display/report the surface reaction rates using the **Kinetic Rate of Reaction-n(Porous)** in the **Reactions...** category of the variable selection drop-down list.

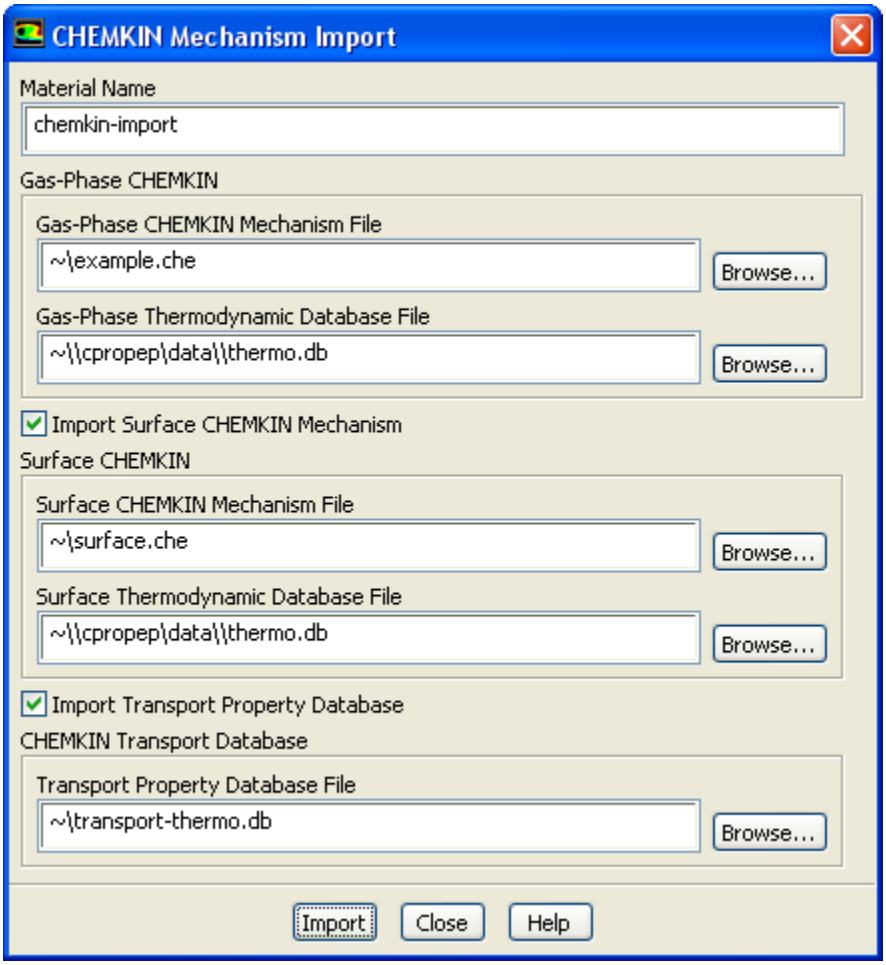
16.2.8. Importing a Surface Kinetic Mechanism in CHEMKIN Format

Importing surface kinetic mechanisms in CHEMKIN format (*Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827)) requires that the gas-phase mechanism file accompany the surface mechanism file for full compatibility with CHEMKIN. If the gas-phase mechanism file is not available, then you will need to create one that you will import along with the surface mechanism file. The mechanism files are imported into ANSYS FLUENT using the **CHEMKIN Mechanism Import** dialog box (*Figure 16.13* (p. 833)).

File → **Import** → **CHEMKIN Mechanism...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.13 The CHEMKIN Mechanism Import Dialog Box for Surface Kinetics



In the **CHEMKIN Mechanism Import** dialog box

1. Enter a name for the chemical mechanism under **Material Name**.
2. Enable **Import Surface CHEMKIN Mechanism**.
3. Enter the path to the **Gas-Phase CHEMKIN Mechanism File** (e.g., *path*/ `gas-file.che`) and the **Surface CHEMKIN Mechanism File** (e.g., *path*/ `surface-file.che`).
4. Specify the location of the **Gas-Phase** and **Surface Thermodynamic Database File**. The thermodynamic database format is detailed in the CHEMKIN User's Guide [*41*] (p. 2211). The default `thermo.db` file supplied with ANSYS FLUENT has only gas-phase species available. You will need to supply a surface `thermo.db` file for your surface species if this `thermo` information is not in the mechanism file.

   Important

   Note that ANSYS FLUENT will initially search for the thermodynamic data in the **Surface CHEMKIN Mechanism File**. If the data does not exist in the mechanism file, then ANSYS FLUENT will search for the thermodynamic data in the specified **Surface Thermodynamic Database File**.

5. (Optional). Import transport properties by enabling the **Import Transport Property Database** and entering the path to that file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To read in KINetics transport properties, go to *Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827) for detailed information.

6. Click the **Import** button.

ANSYS FLUENT will create a material with the specified name, which will contain the CHEMKIN data for the species and reactions, and add it to the list of **Fluent Mixture Materials**. You can view all of the reactions by clicking the **Edit...** button to the right of **Mechanism**, under **Properties** in the *Create/Edit Materials Dialog Box* (p. 1736).

Note that for surface reaction mechanisms, the surface reaction rate constant can be expressed in terms of a sticking coefficient. ANSYS FLUENT will convert this sticking coefficient form to the Arrhenius rate expression [*40*] (p. 2211).

Important

The CHEMKIN import facility does not provide full compatibility with all CHEMKIN rate formulations. To access more complete functionality, you should consider the KINetics module option described in *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801).

16.2.8.1. Compatibility and Limitations for Gas Phase Reactions

ANSYS FLUENT will allow for the following reaction types:

- Arrhenius reactions with arbitrary reaction order, third-body efficiencies and non-integer stoichiometric coefficients.
- Pressure-dependent reactions (Lindemann, Troe and SRI forms)
- Arbitrary reaction units
- Duplicate reactions (keyword DUP)

ANSYS FLUENT will not allow for the following reaction types:

- Landau-Teller reactions (keyword LT)
- Reverse Landau-Teller reactions (keyword RLT)
- Janev reactions (keyword JAN)
- Exponential modified power series reactions (keyword FIT1)
- Radiation reactions (keyword HV)
- Energy loss reactions (keyword EXCI)
- Multi-fluid temperature dependence reactions (keyword TDEP)
- Electron momentum transfer collision frequency (keyword MOME)
- Arbitrary reverse reaction (keyword REV)

Important

Note that the reaction types that ANSYS FLUENT will not allow are mostly applicable to plasmas.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.2.8.2. Compatibility and Limitations for Surface Reactions

ANSYS FLUENT will allow for the following reaction types:

- Arrhenius reactions with arbitrary reaction order, third-body efficiencies and non-integer stoichiometric coefficients.
- Sticking coefficients (keyword STICK). Fluent converts these to an equivalent Arrhenius expression.
- Arbitrary reaction units
- Duplicate reactions (keyword DUP)
- Surface coverage modification (keyword COV)

ANSYS FLUENT will not allow for the following reaction types:

- Ion-Energy Dependent reaction (keyword ENRGDEP)
- Bohm rate expressions (keyword BOHM)
- Ion-Enhanced reaction
- Motz-Wise correction (keywords MWON and MWOFF)

Important

ANSYS FLUENT will warn you of any incompatibilities.

For a detailed description of the keywords, see [*40*] (p. 2211).

16.3. Particle Surface Reactions

As described in The Multiple Surface Reactions Model in the Theory Guide, it is possible to define multiple particle surface reactions to model the surface combustion of a combusting discrete-phase particle. For more information about the theoretical background of particle surface reactions, see Particle Surface Reactions in the Theory Guide. Information about using particle surface reactions is provided in the following subsections:

16.3.1. User Inputs for Particle Surface Reactions
16.3.2. Modeling Gaseous Solid Catalyzed Reactions
16.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion

16.3.1. User Inputs for Particle Surface Reactions

The setup procedure for particle surface reactions requires only a few inputs in addition to the procedure for volumetric reactions described in *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799) – *Defining Other Sources of Chemical Species* (p. 822). These additional inputs are as follows:

- In the **Species Model** dialog box, enable the **Particle Surface** option under **Reactions**.

  Models → **Species** → **Edit...**

- When you specify the species involved in the particle surface reaction, be sure to identify the surface species, as described in *Defining Properties for the Mixture and Its Constituent Species* (p. 805).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

You will find all species (including the surface species) in the list of **Fluent Fluid Materials**. If, for example, you are modeling coal gasification, you will find solid carbon, C(s), in the materials list for the **fluid** material type.

- For each particle surface reaction, select **Particle Surface** as the **Reaction Type** in the **Reactions** dialog box, and specify the following parameters (in addition to those described in *Defining Reactions* (p. 809)):

  Diffusion-Limited Species
  When there is more than one gaseous reactant taking part in the particle surface reaction, the diffusion-limited species is the species for which the concentration gradient between the bulk and the particle surface is the largest. See Figure 7.1: "A Reacting Particle in the Multiple Surface Reactions Model" in the Theory Guide for an illustration of this concept. In most cases, there is a single gas-phase reactant and the diffusion-limited species does not need to be defined.

  Catalyst Species
  This option is available only when there are no solid species defined in the stoichiometry of the particle surface reaction. In such a case, you will need to specify the solid species that acts as a catalyst for the reaction. The reaction will proceed only on the particles that contain this solid specie. See *Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion* (p. 836) for details on defining the particle surface species mass fractions.

  Diffusion Rate Constant
  ($C_{1,r}$ in Equation 7–68 in the Theory Guide)

  Effectiveness Factor
  (η_r in Equation 7–66 in the Theory Guide)

16.3.2. Modeling Gaseous Solid Catalyzed Reactions

The catalytic particle surface reaction option is enabled in ANSYS FLUENT when **Particle Surface** is selected as the **Reaction Type** in the *Reactions Dialog Box* (p. 1765) and there are no solid species in the reaction stoichiometry. The solid species acting as a catalyst for the reaction is defined in the **Reactions** dialog box. The catalytic particle surface reaction will proceed only on those particles containing the catalyst specie.

16.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion

When you use the multiple surface reactions model, the procedure for setting up a problem involving a discrete phase is slightly different from that outlined in *Steps for Using the Discrete Phase Models* (p. 1009). The revised procedure is as follows:

1. Enable any of the discrete phase modeling options, if relevant, as described in *Physical Models for the Discrete Phase Model* (p. 1015).
2. Specify the initial conditions, as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022).
3. Define the boundary conditions, as described in *Setting Boundary Conditions for the Discrete Phase* (p. 1050).
4. Define the material properties, as described in *Setting Material Properties for the Discrete Phase* (p. 1054).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

You must select **multiple-surface-reactions** in the **Combustion Model** drop-down list in the **Create/Edit Materials** dialog box before you can proceed to the next step.

5. If you have defined more than one particle surface species, for example, carbon (**C<s>**) and sulfur (**S<s>**), you will need to return to the **Set Injection Properties** dialog box (or **Set Multiple Injection Properties** dialog box) to specify the mass fraction of each particle surface species in the combusting particle. Click the **Multiple Reactions** tab, and enter the **Species Mass Fractions**. These mass fractions refer to the combustible fraction of the combusting particle, and should sum to 1. If there is only one surface species in the mixture material, the mass fraction of that species will be set to 1, and you will not specify anything under **Multiple Surface Reactions**.
6. Set the solution parameters and solve the problem, as described in *Solution Strategies for the Discrete Phase* (p. 1063).
7. Examine the results, as described in *Postprocessing for the Discrete Phase* (p. 1067).

Important

Solid deposition reactions on the particle are not allowed together with custom laws.

16.4. Species Transport Without Reactions

In addition to the volumetric and surface reactions described in the previous sections, you can also use ANSYS FLUENT to solve a species mixing problem without reactions. The species transport equations that ANSYS FLUENT will solve are described in in the Theory Guide, and the procedure you will follow to set up the non-reacting species transport problem is the same as that described in *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799) – *Defining Other Sources of Chemical Species* (p. 822), with some simplifications.

The basic steps are listed below:

1. Enable **Species Transport** in the **Species Model** dialog box and select the appropriate **Mixture Material**.

   Models → **Species** → **Edit...**

   See *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799) for information about the mixture material concept, and *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801) for more details about using the **Species Model** dialog box.
2. (optional) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option.
3. Check and/or define the properties of the mixture and its constituent species.

   Materials

   Mixture properties include the following:

   - species in the mixture
   - other physical properties (e.g., viscosity, specific heat)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See *Defining Properties for the Mixture and Its Constituent Species* (p. 805) for details.

4. Set species boundary conditions, as described in *Defining Cell Zone and Boundary Conditions for Species* (p. 822).

No special solution procedures are usually required for a non-reacting species transport calculation. Upon completion of the calculation, you can display or report the following quantities:

- **Mass fraction of species-n**
- **Mole fraction of species-n**
- **Concentration of species-n**
- **Lam Diff Coef of species-n**
- **Eff Diff Coef of species-n**
- **Enthalpy of species-n** (pressure-based solver calculations only)
- **Relative Humidity**
- **Mean Molecular Weight**
- **Liquid species mass fraction of species-n** (solidification and melting model only)

These variables are contained in the **Species...** and **Properties...** categories of the variable selection drop-down list that appears in postprocessing dialog boxes. See *Field Function Definitions* (p. 1529) for a complete list of flow variables, field functions, and their definitions. *Displaying Graphics* (p. 1379) and *Reporting Alphanumeric Data* (p. 1509) explain how to generate graphics displays and reports of data.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 17: Modeling Non-Premixed Combustion

In non-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams. This is in contrast to premixed systems, in which reactants are mixed at the molecular level before burning. Examples of non-premixed combustion include pulverized coal furnaces, diesel internal-combustion engines and pool fires.

Under certain assumptions, the thermochemistry can be reduced to a single parameter: the mixture fraction. The mixture fraction, denoted by f, is the mass fraction that originated from the fuel stream. In other words, it is the local mass fraction of burnt and unburnt fuel stream elements (C, H, etc.) in all the species (CO_2, H_2O, O_2, etc.). The approach is elegant because atomic elements are conserved in chemical reactions. In turn, the mixture fraction is a conserved scalar quantity, and therefore its governing transport equation does not have a source term. Combustion is simplified to a mixing problem, and the difficulties associated with closing non-linear mean reaction rates are avoided. Once mixed, the chemistry can be modeled as being in chemical equilibrium with the Equilibrium model, being near chemical equilibrium with the Steady Laminar Flamelet model, or significantly departing from chemical equilibrium with the Unsteady Laminar Flamelet model.

The non-premixed combustion model is presented in the following sections:

17.1. Steps in Using the Non-Premixed Model
17.2. Setting Up the Equilibrium Chemistry Model
17.3. Setting Up the Steady and Unsteady Laminar Flamelet Models
17.4. Defining the Stream Compositions
17.5. Setting Up Control Parameters
17.6. Calculating the Flamelets
17.7. Calculating the Look-Up Tables
17.8. Defining Non-Premixed Boundary Conditions
17.9. Defining Non-Premixed Physical Properties
17.10. Solution Strategies for Non-Premixed Modeling
17.11. Postprocessing the Non-Premixed Model Results

For theoretical background on the non-premixed combustion model, see "Non-Premixed Combustion" in the Theory Guide.

17.1. Steps in Using the Non-Premixed Model

A description of the user inputs for the non-premixed model is provided in the sections that follow.

17.1.1. Preliminaries
17.1.2. Defining the Problem Type
17.1.3. Overview of the Problem Setup Procedure

17.1.1. Preliminaries

Before turning on the non-premixed combustion model, you must enable turbulence calculations in the *Viscous Model Dialog Box* (p. 1645).

Models → **Viscous** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If your model is non-adiabatic, you should also enable heat transfer (and radiation, if required).

Models → **Energy** → **Edit...**

Models → **Radiation** → **Edit...**

Figure 8.7: "Reacting Systems Requiring Non-Adiabatic Non-Premixed Model Approach" in the Theory Guide illustrates the types of problems that must be treated as non-adiabatic.

17.1.2. Defining the Problem Type

Your first task is to define the type of reaction system and reaction model that you intend to use. This includes selection of the following options:

- Non-premixed or partially premixed model option (see *Modeling Partially Premixed Combustion* (p. 905)).
- Equilibrium chemistry model, steady laminar flamelet model, unsteady laminar flamelet model, or diesel unsteady flamelet.
- Adiabatic or non-adiabatic modeling options (see Non-Adiabatic Extensions of the Non-Premixed Model in the Theory Guide).
- Addition of a secondary stream (equilibrium model only).
- Empirically defined fuel and/or secondary stream composition (equilibrium model only).

You can make these model selections using the **Species Model** dialog box (*Figure 17.6* (p. 844)).

Models → **Species** → **Edit...**

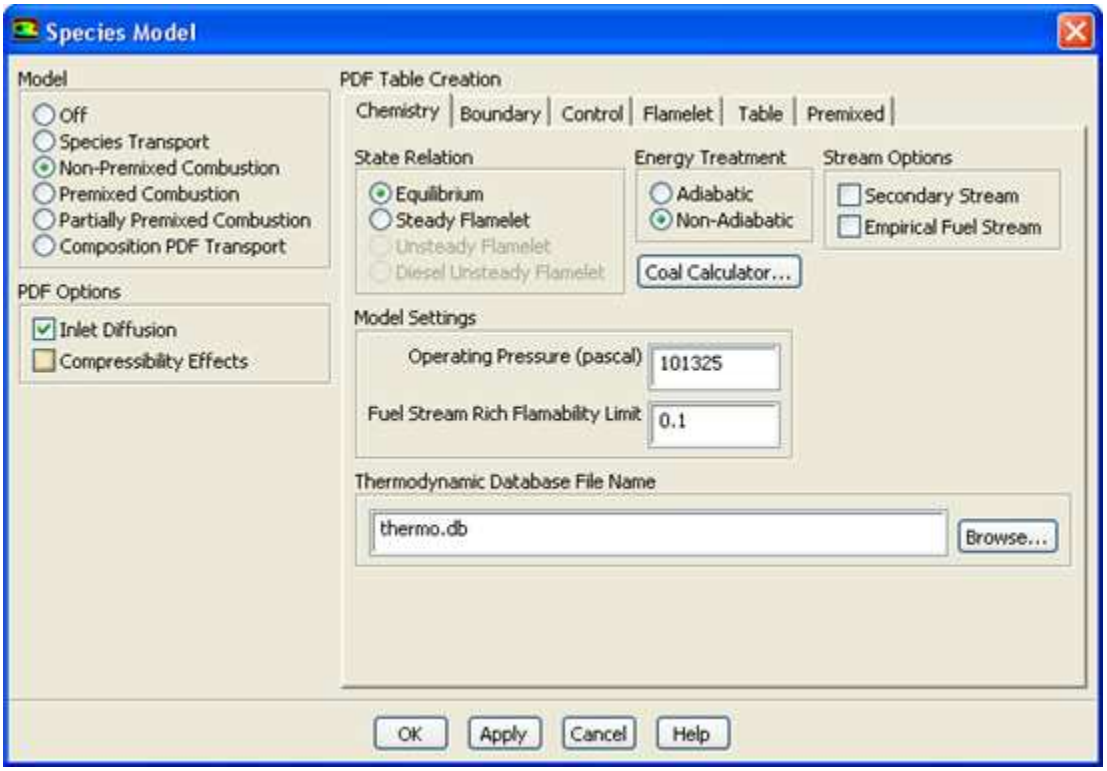
17.1.3. Overview of the Problem Setup Procedure

For a single-mixture-fraction problem, you will perform the following steps:

1. Choose the chemical description of the system: equilibrium, steady flamelet, unsteady flamelet, or diesel unsteady flamelet (*Figure 17.1* (p. 841)).
2. Indicate whether the problem is adiabatic or non-adiabatic.

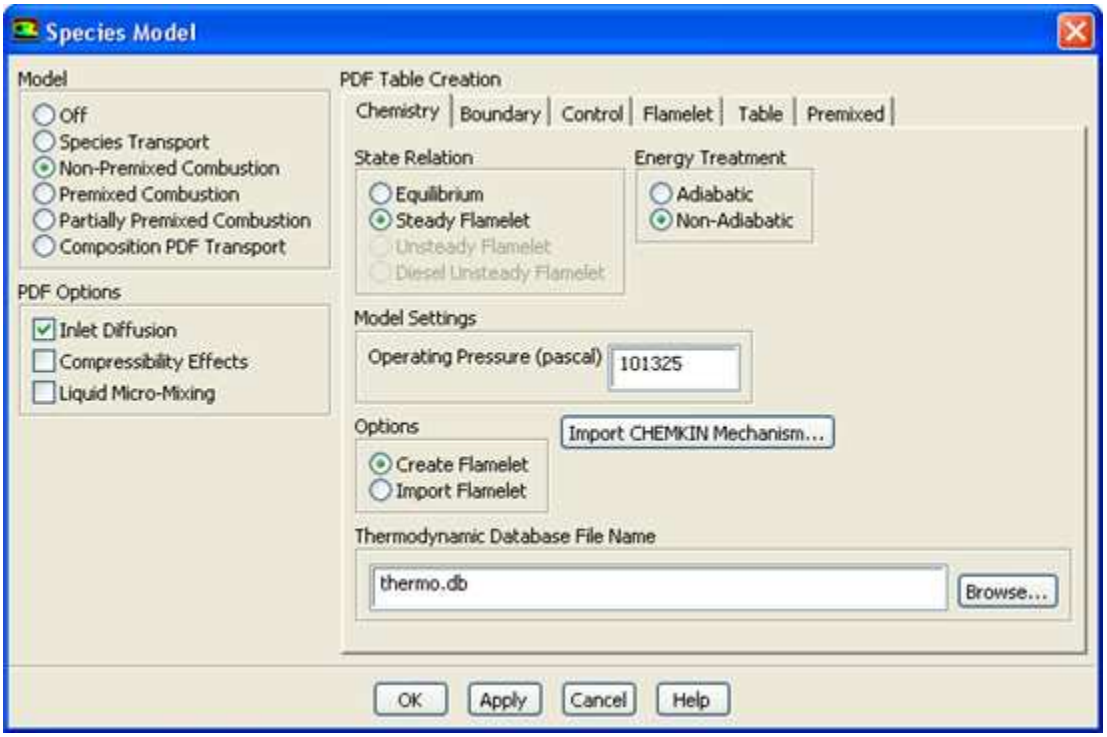
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.1 Defining Equilibrium Chemistry



3. (steady laminar flamelet model only) Import a flamelet file or appropriate CHEMKIN mechanism file if generating flamelets (*Figure 17.2* (p. 841)).

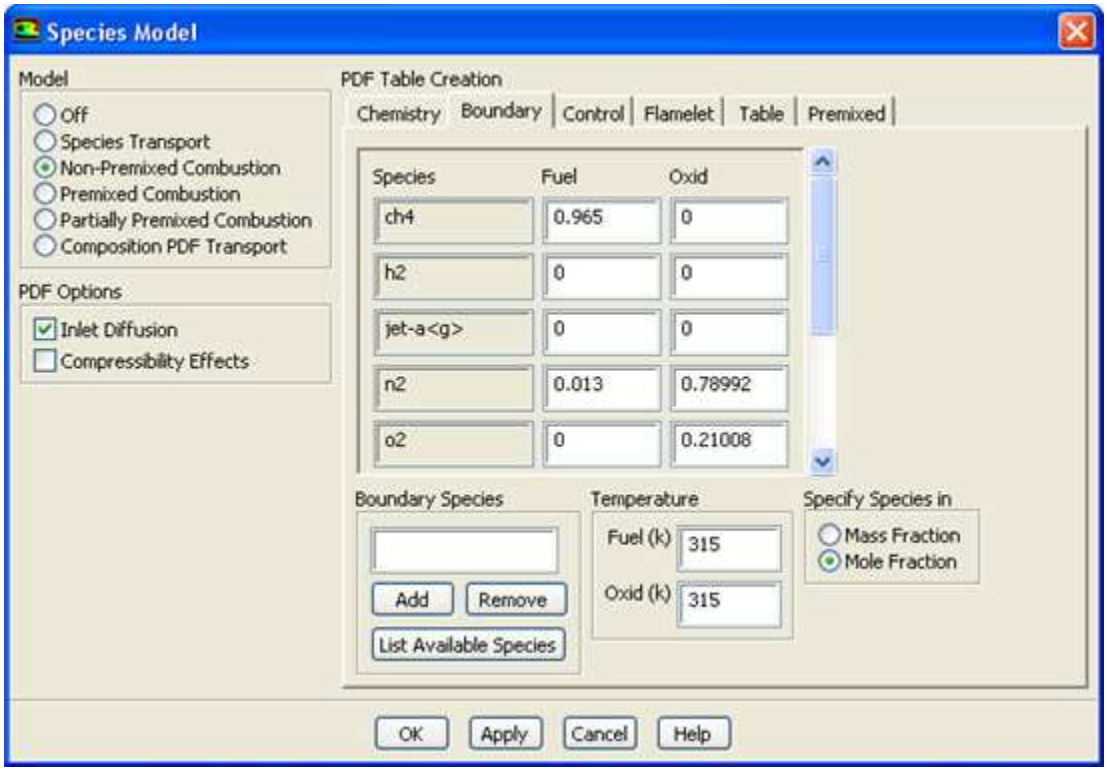
Figure 17.2 Defining Steady Laminar Flamelet Chemistry



4. Define the chemical boundary species to be considered for the streams in the reacting system model. Note that this step is not relevant in the case of flamelet import (*Figure 17.3* (p. 842)).

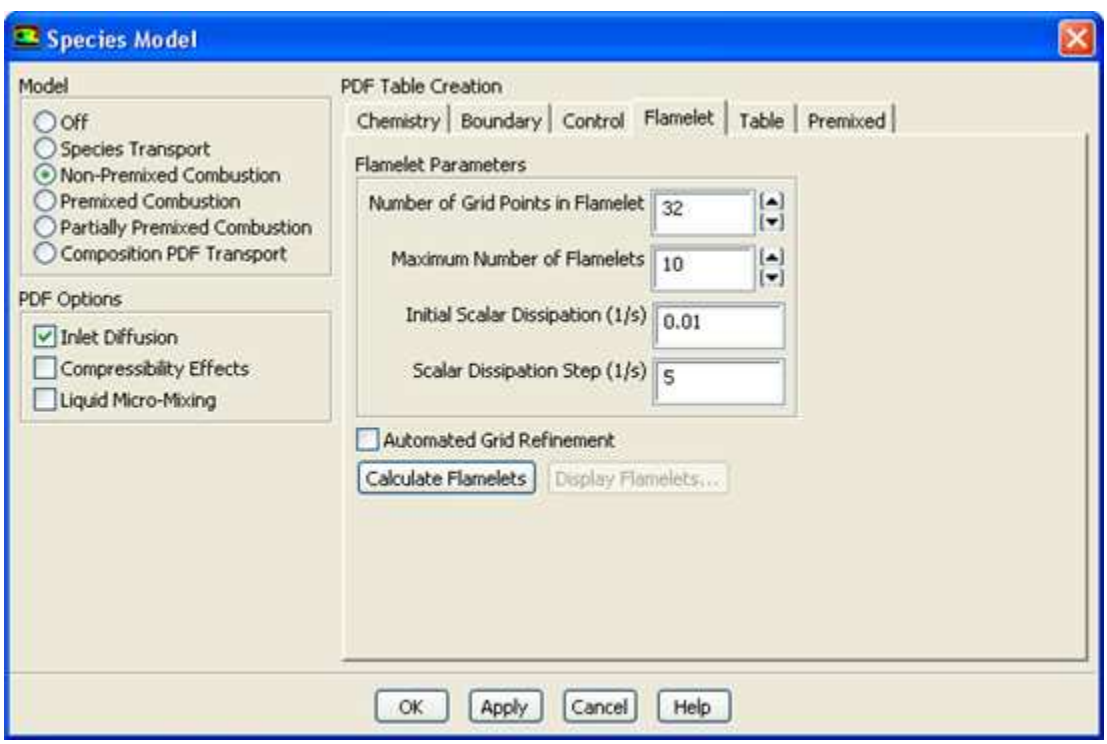
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.3 Defining Chemical Boundary Species



5. (steady laminar flamelet model only) If you are generating flamelets, compute the flamelet state relationships of species mass fractions, density, and temperature as a function of mixture fraction and scalar dissipation (*Figure 17.4* (p. 842)).

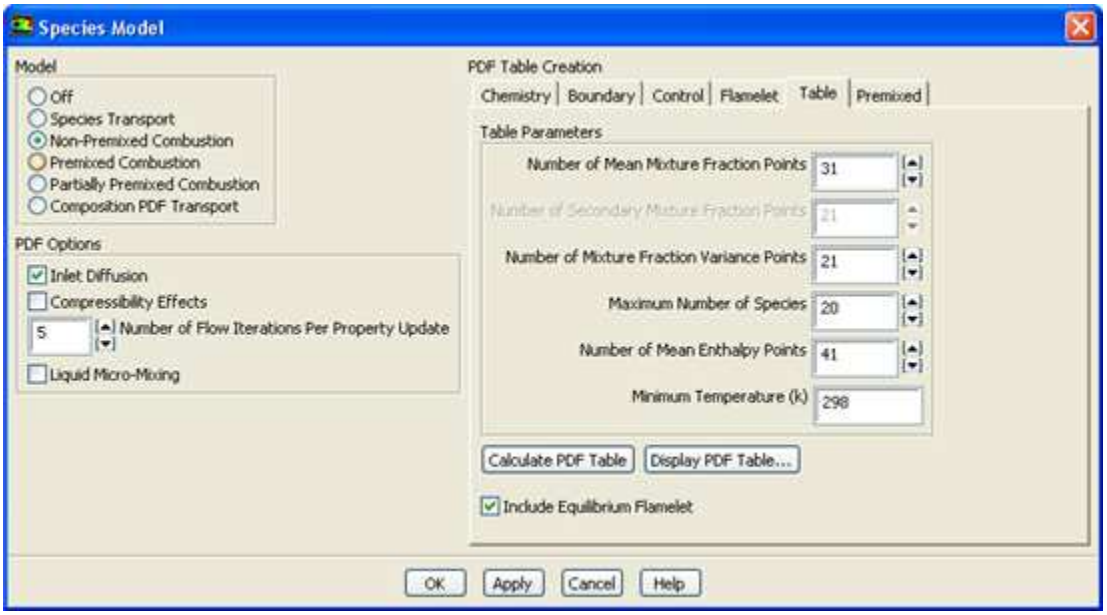
Figure 17.4 Calculating Steady Flamelets



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Compute the final chemistry look-up table, containing mean values of species fractions, density, and temperature as a function of mean mixture fraction, mixture fraction variance, and possibly enthalpy and scalar dissipation. The contents of this look-up table will reflect your preceding inputs describing the turbulent reacting system (*Figure 17.5* (p. 843)).

Figure 17.5 Calculating the Chemistry Look-Up Table



The look-up table is the stored result of the integration of Equation 8–16 (or Equation 8–24) and Equation 8–18 (in the Theory Guide). The look-up table will be used in ANSYS FLUENT to determine mean species mass fractions, density, and temperature from the values of mean mixture fraction ($\overline{f}$), mixture fraction variance ($\overline{f'^2}$), and possibly mean enthalpy ($\overline{H}$) and mean scalar dissipation ($\overline{\chi}$) as they are computed during the ANSYS FLUENT calculation of the reacting flow. See Look-Up Tables for Adiabatic Systems and Figure 8.8: "Visual Representation of a Look-Up Table for the Scalar (Mean Value of Mass Fractions, Density, or Temperature) as a Function of Mean Mixture Fraction and Mixture Fraction Variance in Adiabatic Single-Mixture-Fraction Systems" and Figure 8.10: "Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems" in the Theory Guide.

For a problem that includes a secondary stream (and, therefore, a second mixture fraction), you will perform the first two steps listed above for the single-mixture-fraction approach and then prepare a look-up table of instantaneous properties using Equation 8–12 or Equation 8–14 in the Theory Guide.

17.2. Setting Up the Equilibrium Chemistry Model

In the equilibrium chemistry model, the concentrations of species of interest are determined from the mixture fraction using the assumption of chemical equilibrium (see in the Theory Guide). With this model, you can include the effects of intermediate species and dissociation reactions, producing more realistic predictions of flame temperatures than the Eddy-Dissipation model. When you choose the equilibrium chemistry option, you will have the opportunity to use the rich flammability limit (RFL) option.

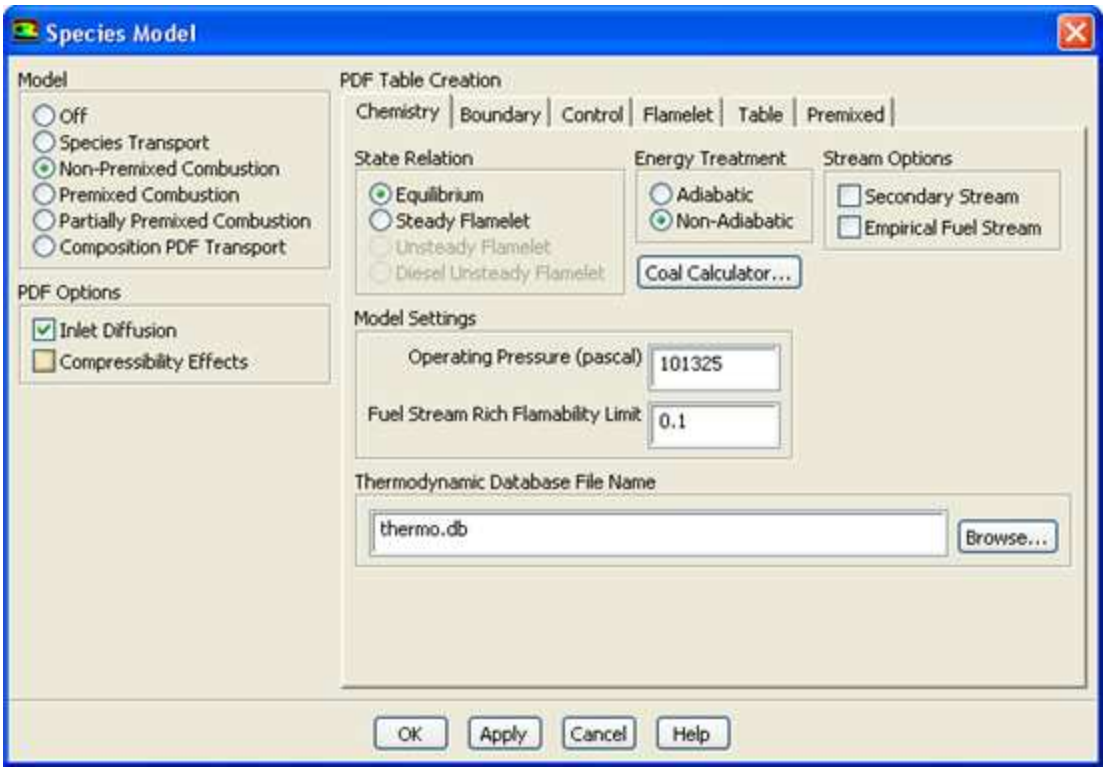
To enable the equilibrium chemistry model

1. Select **Non-Premixed Combustion** in the **Species Model** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select **Equilibrium** in the **Chemistry** tab of the **Species Model** dialog box.

Figure 17.6 The Species Model Dialog Box (Chemistry Tab)



For additional information, please see the following sections:

17.2.1. Choosing Adiabatic or Non-Adiabatic Options
17.2.2. Specifying the Operating Pressure for the System
17.2.3. Enabling a Secondary Inlet Stream
17.2.4. Choosing to Define the Fuel Stream(s) Empirically
17.2.5. Enabling the Rich Flammability Limit (RFL) Option

17.2.1. Choosing Adiabatic or Non-Adiabatic Options

You should use the non-adiabatic modeling option if your problem definition in ANSYS FLUENT will include one or more of the following:

- radiation or wall heat transfer
- multiple fuel inlets at different temperatures
- multiple oxidant inlets at different temperatures
- liquid fuel, coal particles, and/or heat transfer to inert particles

Note that the adiabatic model is a simpler model involving a two-dimensional look-up table in which scalars depend only on $\overline{f}$ and $\overline{f'^2}$ (or on f_{fuel} and p_{sec}). If your model is defined as adiabatic, you will not need to solve the energy equation in ANSYS FLUENT and the system temperature will be determined directly from the mixture fraction and the fuel and oxidant inlet temperatures. The non-adiabatic case will be more complex and more time-consuming to compute, requiring the generation of three-dimensional look-up tables. However, the non-adiabatic model option allows you to include the types of reacting systems described above.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Adiabatic** or **Non-Adiabatic** in the **Chemistry** tab of the **Species Model** dialog box.

17.2.2. Specifying the Operating Pressure for the System

The system **Operating Pressure** is used to calculate density using the ideal gas law. For non-adiabatic simulations, the **Compressibility Effects** under **PDF Options** can be enabled to account for cases where substantial pressure changes occur in time and/or space. In such cases it is assumed that the species mass fractions do not change with pressure, and the density is calculated as

$$\overline{\rho} = \rho_{op} \frac{\overline{p}}{p_{op}} \tag{17–1}$$

where ρ_{op} is the density at the specified **Operating Pressure** (p_{op}), and $\overline{p}$ is the local mean pressure in an ANSYS FLUENT cell.

When the **Compressibility Effects** option is enabled, the flow operating pressure (set in the **Operating Conditions** dialog box) can differ from the Non-Premixed model operating pressure. To distinguish this difference, the **Operating Pressure** name tag in the **Species Model** dialog box changes to **Equilibrium Operating Pressure** when the compressibility effects option is enabled.

See *Solution Strategies for Non-Premixed Modeling* (p. 886) for details about enabling compressibility effects.

17.2.3. Enabling a Secondary Inlet Stream

If you are modeling a system consisting of a single fuel and a single oxidizer stream, you do not need to enable a secondary stream in your PDF calculation. As discussed in Definition of the Mixture Fraction in the Theory Guide, a secondary stream should be enabled if your PDF reaction model will include any of the following:

- two dissimilar gaseous fuel streams
  - In these simulations, the fuel stream defines one of the fuels and the secondary stream defines the second fuel.
- mixed fuel systems of dissimilar gaseous and liquid fuel
  - In these simulations, the fuel stream defines the gaseous fuel and the secondary stream defines the liquid fuel (or vice versa).
- mixed fuel systems of dissimilar gaseous and coal fuels
  - In these simulations, you can use the fuel stream or the secondary stream to define either the coal or the gaseous fuel. See *Modeling Coal Combustion Using the Non-Premixed Model* (p. 855) regarding coal combustion simulations with the non-premixed combustion model.
- mixed fuel systems of coal and liquid fuel
  - In these simulations, you can use the fuel stream or the secondary stream to define either the coal or the liquid fuel. See *Modeling Coal Combustion Using the Non-Premixed Model* (p. 855) regarding coal combustion simulations with the non-premixed combustion model.
- coal combustion
  - Coal combustion can be more accurately modeled by using a secondary stream to track the distinct volatile and char off-gases. The fuel stream must define the char and the secondary stream must define the volatile components of the coal. See *Modeling Coal Combustion Using the Non-Premixed Model* (p. 855) regarding coal combustion simulations with the non-premixed combustion model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- a single fuel with two dissimilar oxidizer streams
  - In these simulations, the fuel stream defines the fuel, the oxidizer stream defines one of the oxidizers, and the secondary stream defines the second oxidizer.

To include a secondary stream in your model, turn on the **Secondary Stream** option under **Stream Options** in the **Chemistry** tab.

Important

Using a secondary stream can substantially increase the calculation time for your simulation since the multi-dimensional PDF integrations are performed at run-time. Alternatively, ANSYS FLUENT can perform a full tabulation of the PDF integrations, as detailed in *Full Tabulation of the Two-Mixture-Fraction Model* (p. 873).

17.2.4. Choosing to Define the Fuel Stream(s) Empirically

The empirical fuel option provides an alternative method for defining the composition of the fuel or secondary stream when the individual species components of the fuel are unknown. In other words, you will define the elemental fraction *not* the individual species. When this option is disabled, you will define the chemical species which are present in each stream and the mass or mole fraction of each species, as described in *Defining the Stream Compositions* (p. 851). The option for defining an empirical fuel stream is particularly useful for coal combustion simulations (see *Modeling Coal Combustion Using the Non-Premixed Model* (p. 855)) or for simulations involving other complex hydrocarbon mixtures.

To define a fuel or secondary stream empirically

1. Turn on the **Empirical Fuel Stream** option under **Stream Options** in the **Chemistry** tab of the **Species Model** dialog box. If you have a secondary stream, enable the **Empirical Secondary Stream** option, or both as appropriate.
2. Specify the appropriate lower heating value (e.g. **Empirical Fuel Lower Caloric Value**, **Empirical Secondary Lower Caloric Value**), specific heat (**Empirical Fuel Specific Heat**, **Empirical Secondary Specific Heat**), and molecular weight (**Empirical Fuel Molecular Weight**, **Empirical Secondary Molecular Weight**) for each empirically defined stream.

Important

The empirical definition option is available only with the full equilibrium chemistry model. It cannot be used with the rich flamability limit (RFL) option or the steady and unsteady laminar flamelet models, since equilibrium calculations are required for the determination of the fuel composition.

Important

The empirical fuel and secondary molecular weights are only required if your empirical streams are entering the domain via an inlet boundary, or if you are using the partially premixed model. If you are using the non-premixed model and the empirically defined streams originate from the dispersed phase (for example, if you are modeling coal or liquid fuel combustion) the molecular weights are not required for the computation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.2.5. Enabling the Rich Flammability Limit (RFL) Option

You can define a rich limit on the mixture fraction when the equilibrium chemistry option is used. Input of the rich limit is accomplished by specifying a value of the **Rich Flammability Limit** for the appropriate **Fuel Stream**, **Secondary Stream**, or both. You will not be allowed to specify the **Rich Flammability Limit** if you have used the empirical definition option for fuel composition.

ANSYS FLUENT will compute the composition at the rich limit using equilibrium. For mixture fraction values above this limit, ANSYS FLUENT will suspend the equilibrium chemistry calculation and will compute the composition based on mixing, but not burning, of the fuel with the composition at the rich limit. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you use a rich limit that is less than 1.0, equilibrium calculations are suspended whenever f, f_{fuel}, or f_{sec} exceeds the limit. This RFL model is often more accurate than the assumption of chemical equilibrium for rich mixtures, and also avoids complex equilibrium calculations, speeding up the preparation of the look-up tables. An RFL value of approximately twice the stoichiometric mixture fraction is appropriate.

For the **Secondary Stream**, the rich flammability limit controls the equilibrium calculation for the secondary mixture fraction. If your secondary stream is not a fuel, you should use an RFL value of 1. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you input a rich limit that is less than 1.0, equilibrium calculations are suspended whenever f_{sec} exceeds the limit. (Note that it is the secondary mixture fraction f_{sec} and not the partial fraction p_{sec} that is used here.)

Important

Experimental studies and reviews [*11*] (p. 2209), [*80*] (p. 2213) have shown that although the fuel-lean flame region approximates thermodynamic equilibrium, non-equilibrium kinetics will prevail under fuel-rich conditions. Therefore, for non-empirically defined fuels, the RFL model is strongly recommended.

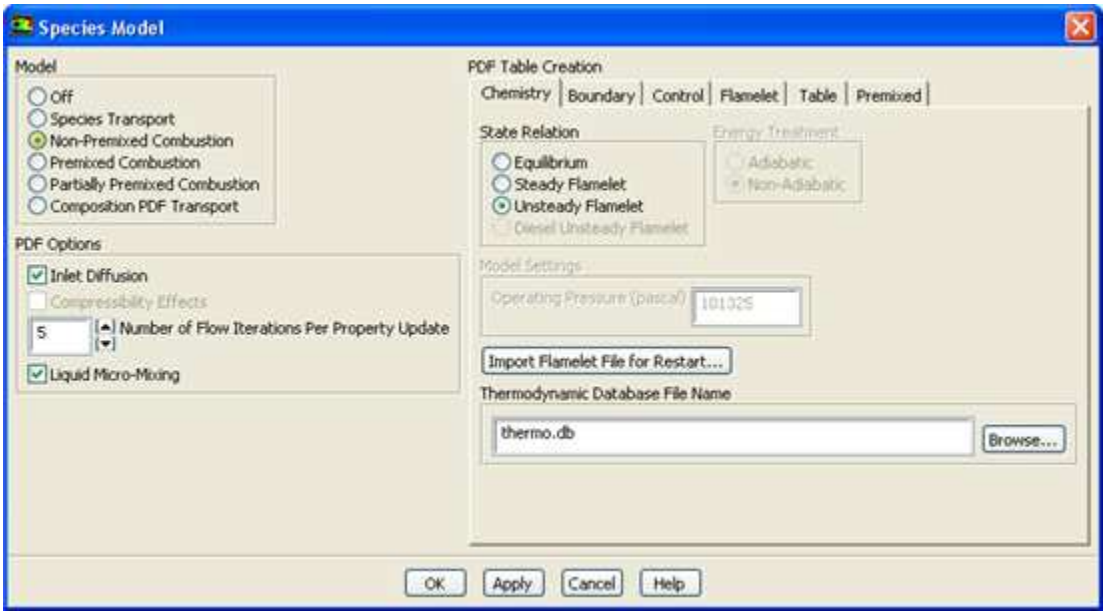
17.3. Setting Up the Steady and Unsteady Laminar Flamelet Models

To enable the laminar flamelet models

1. Select **Non-Premixed Combustion** in the **Species Model** dialog box.
2. Select **Steady Flamelet** or **Unsteady Flamelet** in the **Chemistry** tab of the **Species Model** dialog box. See *Using the Unsteady Laminar Flamelet Model* (p. 850).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.7 The Chemistry Tab for the Unsteady Flamelet Model



For additional information, please see the following sections:

17.3.1. Choosing Adiabatic or Non-Adiabatic Options
17.3.2. Specifying the Operating Pressure for the System
17.3.3. Specifying a Chemical Mechanism File for Flamelet Generation
17.3.4. Importing a Flamelet
17.3.5. Using the Unsteady Laminar Flamelet Model
17.3.6. Using the Diesel Unsteady Laminar Flamelet Model

17.3.1. Choosing Adiabatic or Non-Adiabatic Options

Select **Adiabatic** or **Non-Adiabatic** in the **Chemistry** tab of the **Species Model** dialog box. See the discussion in *Choosing Adiabatic or Non-Adiabatic Options* (p. 844) about the two options.

17.3.2. Specifying the Operating Pressure for the System

The system **Operating Pressure** is used to calculate density using the ideal gas law. When the **Compressibility Effects** option is enabled, the name **Operating Pressure** is changed to **Equilibrium Operating Pressure** since the non-premixed combustion model operating pressure can differ from the flow operating pressure. *Specifying the Operating Pressure for the System* (p. 845) provides more information about this value.

You can use the steady or unsteady laminar flamelet model for reactions in liquid systems. To do so, enable **Liquid Micro-Mixing** under **PDF Options**. The **Liquid Micro-Mixing** option is discussed in detail in Liquid Reactions in the Theory Guide.

17.3.3. Specifying a Chemical Mechanism File for Flamelet Generation

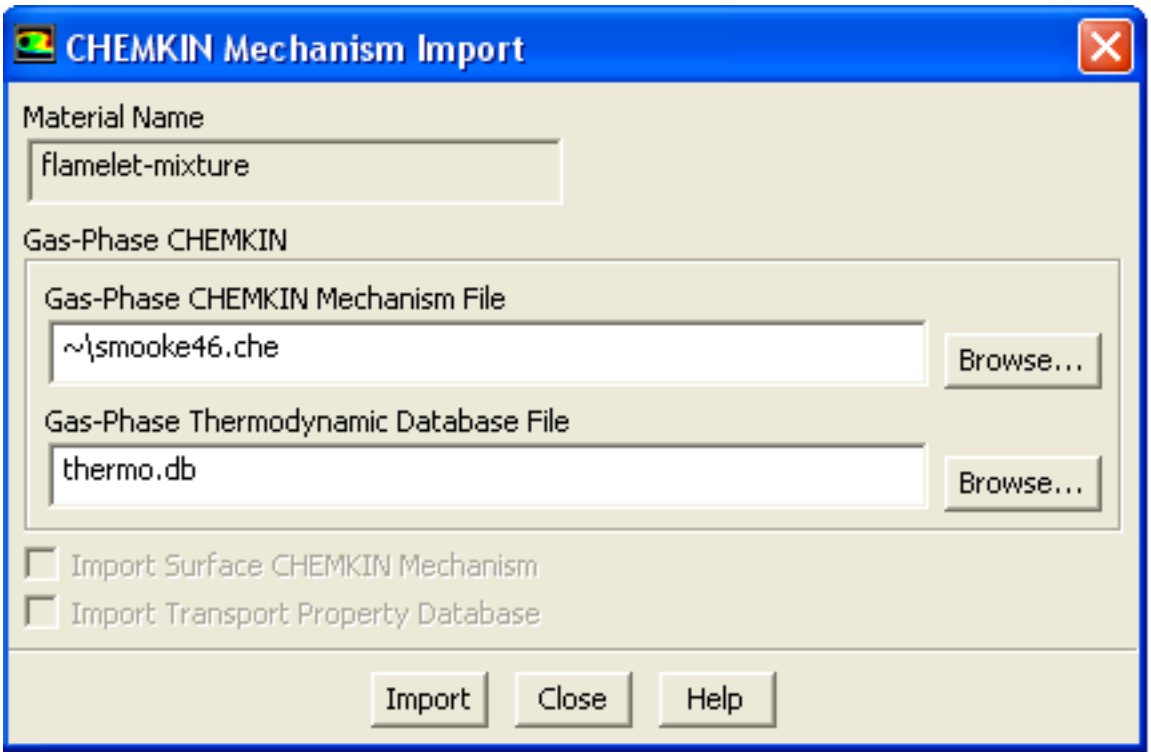
If you are generating a flamelet file yourself, you will need to read in the chemical kinetic mechanism and thermodynamic data. The mechanism and thermodynamic data must be in CHEMKIN format [*41*] (p. 2211).

To read in a CHEMKIN mechanism, select the **Create Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box and click the **Import CHEMKIN Mechanism...** button. When you click this button, the **CHEMKIN Mechanism Import** dialog box (*Figure 17.8* (p. 849)) will open. In the **CHEMKIN Mechanism Import**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog box, enter the path to the CHEMKIN file to be read under **Gas-Phase CHEMKIN Mechanism File** and specify the location of the thermodynamic database under **Gas-Phase Thermodynamic Database File**. Alternatively, you can click the appropriate **Browse...** button to open *The **Select File** Dialog Box* (p. 33), or simply use the default `thermo.db` which is already provided.

Figure 17.8 The Laminar Flamelet CHEMKIN Mechanism Import Dialog Box



Click **Import** in the **CHEMKIN Mechanism Import** dialog box to read the specified files into ANSYS FLUENT. Note that the import is limited to mechanisms with 300 or fewer species and 1500 or fewer reactions.

17.3.4. Importing a Flamelet

To import an existing flamelet file

1. Select the **Import Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box.
2. (steady flamelet only) Select either **Standard**, **Oppdif** or **CFX-RIF** format under **File Type**.
3. (steady flamelet only) If you selected **Oppdif** as the **File Type**, choose a **Mixture Fraction Method**. Select **Drake** if you want to calculate the mixture fraction using carbon and hydrogen elements. Select **Bilger** to calculate the mixture fraction using hydrocarbon formula. Select **Nitrogen** to calculate the mixture fraction in terms of nitrogen species.
4. (steady flamelet only) If you selected the **Oppdif File Type**, you have a choice of importing **Single** or **Multiple** OPPDIF files under **Oppdif Flamelet Type**.
5. Click the **Import Flamelet File...** button. In *The **Select File** Dialog Box* (p. 33), select the file (for a single flamelet) or files (for multiple flamelets) to be read in to ANSYS FLUENT.

After you have completed this step, you can skip ahead to the **Table** tab of the **Species Model** dialog box (see *Calculating the Look-Up Tables* (p. 871)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.3.5. Using the Unsteady Laminar Flamelet Model

The unsteady laminar flamelet model can only be enabled from a valid steady-state, steady laminar flamelet solution. When enabled, the unsteady laminar flamelet model will change this case to unsteady and post-process a marker probability equation on the frozen flow field. You should hence ensure that the starting steady-state, steady laminar flamelet solution is fully converged.

When the **Unsteady Flamelet** is enabled in the **Chemistry** tab, the **Import Flamelet File for Restart...** button appears in the dialog box, allowing you to run the simulation from a previously saved case, data and unsteady flamelet file.

The Unsteady Laminar Flamelet Model requires four user inputs in the **Flamelet** tab:

- The **Number of Grid Points in Flamelet**.
- The **Mixture Fraction Lower Limit for Initial Probability**. The initial condition of the marker probability field is unity for all mean mixture fractions above the **Mixture Fraction Lower Limit for Initial Probability**, and zero for mean mixture fractions below it. Note that this should be specified to be greater than the stoichiometric mixture fraction.
- **Maximum Scalar Dissipation**. Laminar flamelets may extinguish at high scalar dissipations because diffusion in the flamelet overwhelms reaction. It is possible to have unrealistically high modeled scalar dissipation in the 2D or 3D ANSYS FLUENT simulations, which gets transferred to the 1D unsteady flamelet. In order to avoid excessive diffusion in the 1D unsteady flamelet, the instantaneous scalar dissipation in the 1D flamelet is limited to the specified **Maximum Scalar Dissipation**.
- **Courant Number**. The time step for the unsteady probability marker equation is calculated automatically by ANSYS FLUENT based on the **Courant Number**. Larger values imply fewer time steps to convect/diffuse the marker probability out of the domain, but also results in a larger numerical error. The **Courant Number** should be small enough so that the unsteady flamelet mean mass fractions are unchanged with any smaller **Courant Number**. The default value of 1 should be sufficient for most applications.

When these inputs have been set, clicking the **Initialize Unsteady Flamelet Probability** button initializes the marker probability equation, automatically enabling the **Unsteady** solver, while disabling all equations except the **Unsteady Flamelet Probability** equation in the **Solution Controls** task page. This initialization in the **Flamelet** tab also sets the **Time Step Size** in the **Run Calculation** task page.

Important

Do not initialize your solution using the **Solution Initialization** task page. Note that you are postprocessing a probability field on the frozen steady-state flow field, and by clicking the **Initialize Unsteady Flamelet Probability** button, you have already initialized the probability marker field.

Important

If you disable the **Unsteady Laminar Flamelet** model and you want to revert to solving a steady laminar flamelet simulation, make sure you enable **Steady** in the **General** task page and enable all the equations in the **Solution Controls** task page.

17.3.6. Using the Diesel Unsteady Laminar Flamelet Model

The diesel unsteady laminar flamelet model can only be enabled when conditions for compression-ignition are met:

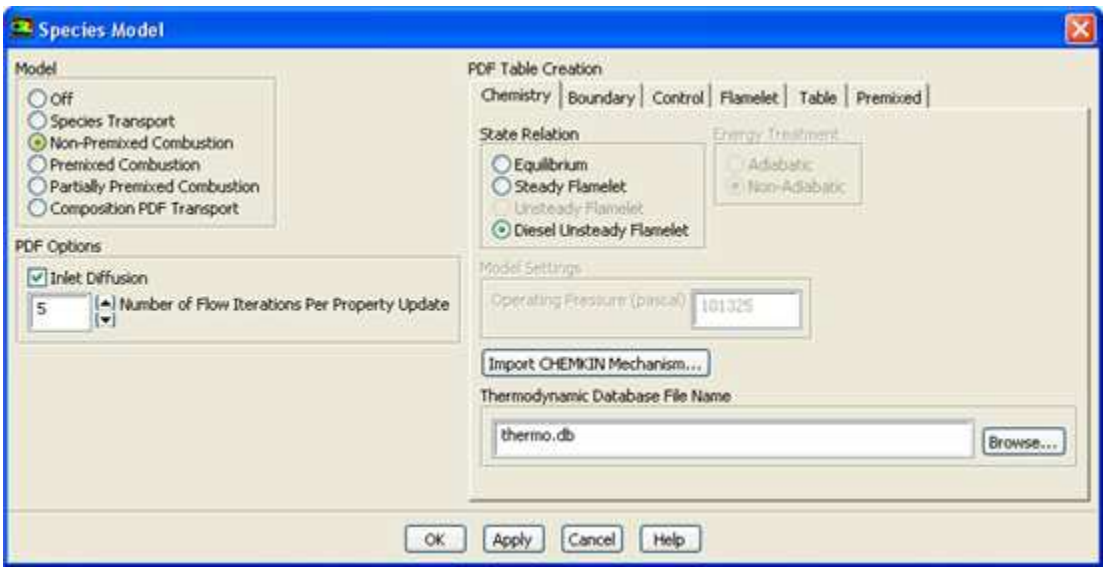
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Transient** solver is selected in the **General** task page.
- The **In-Cylinder** dynamic mesh is enabled.
- The Discrete Phase model option **Interaction with Continuous Phase** is on.

A detailed chemical mechanism is required which should contain kinetic reactions appropriate for compression ignition. The mechanism can include pollutant formation reactions as well if you are interested in modeling emissions.

In the **Chemistry** tab, select **Diesel Unsteady Flamelet**.

Figure 17.9 The Enabled Diesel Unsteady Flamelet Model



With the **Diesel Unsteady Flamelet** model enabled, you can define the stream compositions (*Defining the Stream Compositions* (p. 851)) and the flamelet controls (*Defining the Flamelet Controls* (p. 863)). Note that while you can enter the **Number of Grid Points in Flamelet** in the **Flamelet** tab (*Unsteady Flamelet* (p. 866)), you will not be able to calculate the flamelet or PDF table (*Calculating the Look-Up Tables* (p. 871)). The reason for this is because the table creation and the flamelet calculation are performed at every time step of the ANSYS FLUENT simulation, and not just in a pre-processing step as in other non-premixed combustion models. However, you can set the table parameters in the **Table** tab.

17.4. Defining the Stream Compositions

In ANSYS FLUENT, you will input only the boundary species (i.e., the fuel, oxidizer, and if necessary, secondary stream species). The intermediate and product species will be determined automatically.

ANSYS FLUENT provides you with an initial list of common boundary species (`ch4`, `h2`, `jet-a<g>`, `n2` and `o2`). If your fuel and/or oxidizer is composed of different species, you can add them to the boundary **Species** list. All boundary species must exist in the chemical database and you must enter their names in the same format used in the database, otherwise an error message will be issued.

After defining the boundary species that will be considered in the reaction system, you must define their mole or mass fractions at the fuel and oxidizer inlets and at the secondary inlet, if one exists. (If you choose to define the fuel or secondary stream composition empirically, you will instead enter the parameters described at the end of this section.) For the example shown in Figure 8.12: "Chemical Systems That Can Be Modeled

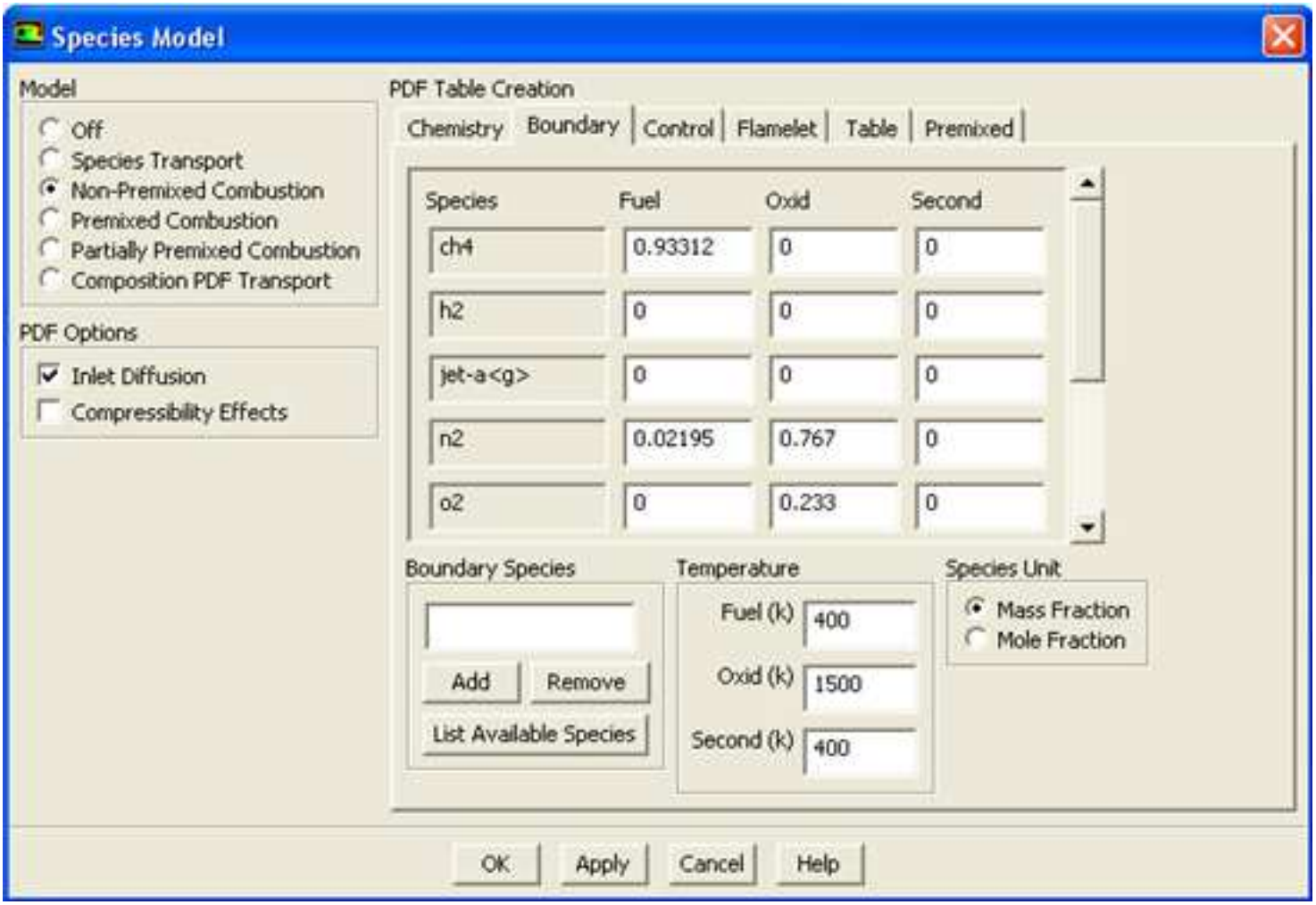
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using a Single Mixture Fraction" in the Theory Guide, for example, the fuel inlet consists of 60% CH_4, 20% CO, 10% CO_2, and 10% C_3H_8.

Finally, the inlet stream temperatures of your reacting system are required for construction of the look-up table and computation of the equilibrium chemistry model.

For the equilibrium chemistry model, the species names are entered using the **Boundary** tab in the **Species Model** dialog box (*Figure 17.10* (p. 852)). If you are generating a steady or unsteady laminar flamelet, the list of boundary species will be automatically filled as all the species in the CHEMKIN mechanism, and you will be unable to change these.

Figure 17.10 The Species Model Dialog Box (Boundary Tab)



The steps for adding new species and defining their compositions is as follows:

1. (equilibrium chemistry model only) If your fuel, oxidizer, or secondary streams are composed of species other than the default species list, type the chemical formula (e.g., `so2` or `S02` for SO_2) under **Boundary Species** and click **Add**. The species will be added to the **Species** list. Continue in this manner until all of the boundary species you want to include are shown in the **Species** list.

   To remove a species from the list, type the chemical formula under **Boundary Species** and click **Remove**. To print a list of all species in the thermodynamic database file (`thermo.db`) in the console window, click **List Available Species**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under **Species Unit**, specify whether you want to enter the **Mass Fraction** or **Mole Fraction**. **Mass Fraction** is the default.
3. For each relevant species in the **Species** list, specify its mass or mole fraction for each stream (**Fuel**, **Oxid**, or **Second** as appropriate) by entering values in the table. Note that if you change from **Mass Fraction** to **Mole Fraction** (or vice versa), all values will be automatically converted if they sum to 0 or 1, so be sure that you are entering either all mass fractions or all mole fractions as appropriate. If the values do not sum to 0 or 1, an error will be issued.
4. Under **Temperature**, specify the following inputs:

   Fuel
   is the temperature of the fuel inlet in your model. In adiabatic simulations, this input (together with the oxidizer inlet temperature) determines the inlet stream temperatures that will be used by ANSYS FLUENT. In non-adiabatic systems, this input should match the inlet thermal boundary condition that you will use in ANSYS FLUENT (although you will enter this boundary condition again in the ANSYS FLUENT session). If your ANSYS FLUENT model will use liquid fuel or coal combustion, define the inlet fuel temperature as the temperature at which vaporization/devolatilization begins (i.e., the **Vaporization Temperature** specified for the discrete-phase material—see *Setting Material Properties for the Discrete Phase* (p. 1054)). For such non-adiabatic systems, the inlet temperature will be used only to adjust the look-up table grid (e.g., the discrete enthalpy values for which the look-up table is computed). Note that if you have more than one fuel inlet, and these inlets are not at the same temperature, you must define your system as non-adiabatic. In this case, you should enter the fuel inlet temperature as the value at the dominant fuel inlet.

   Oxid
   is the temperature of the oxidizer inlet in your model. The issues raised in the discussion of the input of the fuel inlet temperature (directly above) pertain to this input as well.

   Second
   is the temperature of the secondary stream inlet in your model. (This item will appear only when you have defined a secondary inlet.) The issues raised in the discussion of the input of the fuel inlet temperature (directly above) pertain to this input as well.

For additional information, please see the following sections:

17.4.1. Setting Boundary Stream Species
17.4.2. Modifying the Database
17.4.3. Composition Inputs for Empirically-Defined Fuel Streams
17.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model
17.4.5. Modeling Coal Combustion Using the Non-Premixed Model

17.4.1. Setting Boundary Stream Species

In combustion, a large number of intermediate and product species may be produced from a small number of initial boundary species. In ANSYS FLUENT you need to input only the species composition of your boundary species in the fuel, oxidizer, and (if appropriate) secondary streams. ANSYS FLUENT will calculate all intermediate and product species automatically. The following suggestions may be helpful in the definition of the system chemistry:

- For coal combustion, char in the coal should be represented by C(s).

  Important

  Care should be taken to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should be selected only if you are using the empirically-defined input method.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If your fuel composition is known empirically (e.g., $C_{0.9}H_3O_{0.2}$), use the option for an empirically-defined stream (see below).
- If you wish to include the sulfur that may be present in a hydrocarbon fuel, note that this may hinder the convergence of the equilibrium solver, especially if the concentration of sulfur is small. It is therefore recommended that you include sulfur in the calculation only if it is present in considerable quantities.

17.4.1.1. Including Condensed Species

In addition to gaseous species, liquid and solid species can be included in the chemistry calculations. They are often indicated by an "l" or an "s" in parentheses after the species name. If you add a condensed species to the equilibrium chemical system, its density will be retrieved from ANSYS FLUENT's chemical property database file `propdb.scm` if you are using the thermodynamic database file `thermo.db` that is also supplied with ANSYS FLUENT. If you are using a custom thermodynamic database file and want to include a condensed species in the equilibrium system that does not exist in `propdb.scm`, a density of 1000 kg/m^3 will be assumed. The condensed species density can be changed in the *Create/Edit Materials Dialog Box* (p. 1736) after the PDF table has been calculated. If you modify the condensed species density in this manner, you will then need to recalculate the PDF table.

17.4.2. Modifying the Database

If you want to include a new species in your reacting system that is not available in the chemical database, you can add it to the database file, `thermo.db`. The format for `thermo.db` is detailed in [*41*] (p. 2211). If you choose to modify the standard database file, you should create copies of the original file.

17.4.3. Composition Inputs for Empirically-Defined Fuel Streams

As mentioned in *Defining the Problem Type* (p. 840), you can define the composition of a fuel stream (i.e., the standard fuel or a secondary fuel) empirically. For an empirically-defined stream, you will need to enter the atomic mass or mole fractions in addition to the inputs for lower caloric (heating) value of the fuel and the mean specific heat of the fuel that were described previously.

The heat of formation of an empirically defined stream is calculated from the heating value and the atomic composition. The fuel inlet temperature and fuel specific heat are used to calculate the sensible enthalpy. The molecular weight is used for the computation of the unburnt stream density. Note that the unburnt density is only required if the stream enters via an inlet boundary, or if you are using the partially-premixed model.

When an empirically-defined fuel or secondary stream is specified in the **Chemistry** tab (equilibrium chemistry model only) of the **Species Model** dialog box, you must specify the atom fractions of C, H, O, N, and S in that stream instead of the species mass or mole fractions. To avoid confusion, the species fraction inputs for an empirically-defined stream will be grayed out in the table within the **Boundary** tab, leaving only the fields for atom fractions (i.e., **c**, **h**, **o**, **n**, and **s**).

17.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model

Liquid fuel combustion can be modeled with the discrete phase and non-premixed models. In ANSYS FLUENT, the fuel vapor, which is produced by evaporation of the liquid fuel, is defined as the fuel stream. (See *Defining the Stream Compositions* (p. 851).) The liquid fuel that evaporates within the domain appears as a source of the mean fuel mixture fraction.

Within ANSYS FLUENT, you define the liquid fuel discrete-phase model in the usual way. The gas phase (oxidizer) flow inlet is modeled using an inlet mixture fraction of zero and the fuel droplets are introduced

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as discrete phase injections (see *Setting Initial Conditions for the Discrete Phase* (p. 1022)). The property inputs for the liquid fuel droplets are unaltered by the non-premixed model (see *Setting Material Properties for the Discrete Phase* (p. 1054)). Note that when you are requested to input the gas phase species destination for the evaporating liquid, you should input the species that comprises the evaporating stream.

If the fuel stream was defined as a mixture of components, you should select the largest of these components as the "evaporating species". ANSYS FLUENT will ensure that the mass evaporated from the liquid droplet enters the gas phase as a source of the fuel mixture that you defined. The evaporating species you select here is used only to compute the diffusion controlled driving force in the evaporation rate.

17.4.5. Modeling Coal Combustion Using the Non-Premixed Model

If your model involves coal combustion, the fuel and secondary stream compositions can be input in one of several ways. You can use a single mixture fraction (fuel stream) to represent the coal, defining the fuel composition as a mixture of volatiles and char (solid carbon). Alternatively, you can use two mixture fractions (fuel and secondary streams), defining the volatiles and char separately. In two-mixture-fraction models for coal combustion, the fuel stream represents the char and the secondary stream represents volatiles. This section describes the modeling options and special input procedures for coal combustion models using the non-premixed approach.

There are three options for coal combustion:

- When coal is the only fuel in the system, you can model the coal using two mixture fractions, where the primary stream represents the char and the secondary stream represents the volatiles. Generally, the char stream composition is defined as 100% C(s). The volatile stream composition is defined by selecting appropriate species and setting their mole or mass fractions. Alternatively, you can use the empirical method (input of atom fractions) for defining these compositions.

  Important

  Using two mixture fractions to model coal combustion is more accurate than using one mixture fraction as the volatile and char streams are modeled separately. However, the two-mixture-fraction model incurs significant additional computational expense since the multi-dimensional PDF integrations are performed at run-time.

- When coal is the only fuel in the system, you can choose to model the coal using a single mixture fraction (the fuel stream). When this approach is adopted, the fuel composition you define includes both volatile species and char. Char is typically represented by including C(s) in the species list. You can define the fuel stream composition by selecting appropriate species and setting their mole fractions, or by using the empirical method (input of atom fractions). Definition of the composition is described in detail below.

  Important

  Using a single mixture fraction for coal combustion is less accurate than using two mixture fractions. However, convergence in ANSYS FLUENT should be substantially faster than the two-mixture-fraction model.

- When coal is used with another (gaseous or liquid) fuel of different composition, you must model the coal with one mixture fraction and use a second mixture fraction to represent the second (gaseous or

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

liquid) fuel. The stream associated with the coal composition is defined as detailed below for single-mixture-fraction models.

17.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models

When coal is modeled using a single mixture fraction (the fuel stream), the fuel stream composition can be input using the conventional approach or the empirical fuel approach.

- Conventional approach:

  To use the conventional approach, you will need to define the mixture of species in the coal and their mole or mass fractions in the fuel stream. Use the **Boundary** tab in the **Species Model** dialog box to input the list of species (e.g., C_3H_8, CH_4, CO, CO_2, C(s)) that approximate the coal composition, and their mole or mass fractions.

  Note that C(s) is used to represent the char content of the coal. For example, consider a coal that has a molar composition of 40% volatiles and 60% char on a dry ash free (DAF) basis. Assuming the volatiles can be represented by an equimolar mixture of C_3H_8 and CO, the fuel stream composition defined in the **Boundary** tab would be C_3H_8=0.2, CO = 0.2, and C(s)=0.60. Note that the coal composition should always be defined on an ash-free basis, even if ash will be considered in the ANSYS FLUENT calculation.

  To define ash properties, go to the **Create/Edit Materials** dialog box and select **combusting-particle** as the **Material Type**.

  The following table illustrates the conversion from a typical mass-based proximate analysis to the species fraction inputs required by ANSYS FLUENT. Note that the conversion requires that you make an assumption regarding the species representing the volatiles. Here, the volatiles are assumed to exist as an equimolar mix of propane and carbon monoxide.

Proximate Analysis	Weight %	Mass Fraction (DAF)	Moles (DAF)	Mole Fraction (DAF)
Volatiles	30			
$-C_3H_8$		0.2035	0.004625	0.07134
– CO		0.1295	0.004625	0.07134
Fixed Carbon (C(s))	60	0.667	0.05558	0.85732
Ash	10	-	-	-
(Total)			0.06483	1.0

  Moisture in the coal can be considered by adding it in the fuel composition as liquid water, H_2O(l). The moisture can also be defined as water vapor, H_2O, provided that the corresponding latent heat is included in the discrete phase material inputs in ANSYS FLUENT. If the liquid water is used as a boundary species, it should be removed from the list of excluded species (see *Forcing the Exclusion and Inclusion of Equilibrium Species* (p. 862)).

  Important

  Note that if water is included in the coal, the water release is not modeled as evaporation, which is typically the case in the wet combustion model, described in *Particle Types* (p. 1025).

- Empirical fuel approach:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use the empirical approach, enable the **Empirical Fuel Stream** option in the **Chemistry** tab. This method is ideal if you have an elemental analysis of the coal.

In the **Chemistry** tab, input the lower heating value and mean specific heat of the coal. ANSYS FLUENT will use these inputs to determine the mole fractions of the chemical species you have included in the system. Then, in the **Boundary** tab, define the atom fractions of C, H, N, S, and O in the fuel stream.

Note that for both of these composition input methods, you should take care to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should only be selected if you are using the empirical fuel input method.

See *Additional Coal Modeling Inputs in ANSYS FLUENT* (p. 858) for details about further inputs for modeling coal combustion.

17.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models

You can model coal using the two mixture fractions model, where the primary stream represents the char and the secondary stream represents the volatiles.

As in single-mixture-fraction cases, the fuel stream and secondary stream compositions in a two-mixture-fraction case can be input using either the conventional approach or the empirical fuel approach.

- Conventional approach:

  To use the conventional approach, you will need to define the mixture of species in the coal and their mole or mass fractions in the fuel and secondary streams.

  Use the **Boundary** tab of **Species Model** dialog box to define the mole or mass fractions of volatile species in the secondary stream (e.g., C_3H_8, CH_4, CO, CO_2, C(s)). Next, define the mole or mass fractions of species used to represent the char. Generally, you will input 100% C(s) for the fuel stream.

- Empirical fuel approach:

  To use the empirical fuel approach, enable the **Empirical Secondary Stream** option in the **Chemistry** tab for the volatile (secondary) stream. This method is ideal if you have an elemental analysis of the coal.

  In the **Chemistry** tab, input the lower heating value and mean specific heat of the coal. Then, in the **Boundary** tab, define the mole or mass fractions of species used to represent the char. Generally, you will input 100% C(s) for the fuel stream. Finally, define the atom fractions of C, H, N, S, and O in the volatiles. ANSYS FLUENT will use these inputs to determine the mole fractions of the chemical species you have included in the system. For example, consider coal with the following DAF (dry ash free) data and elemental analysis:

Proximate Analysis	**Wt % (dry)**	**Wt % (DAF)**
Volatiles	28	30.4
Char (C(s))	64	69.6
Ash	8	-

Element	**Wt % (DAF)**	**Wt % (DAF)**
C	89.3	89.3

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Wt % (DAF)	Wt % (DAF)
H	5.0	5.0
O	3.4	3.4
N	1.5	2.3
S	0.8	-

(Note that in the final column, for modeling simplicity, the sulfur content of the coal has been combined into the nitrogen mass fraction.)

You can combine the proximate and ultimate analysis data to yield the following elemental composition of the volatile stream:

Element	Mass	Mass Fraction	Moles	Mole Fraction
C	(89.3 - 69.6)	0.65	5.4	0.24
H	5.0	0.16	16	0.70
O	3.4	0.11	0.7	0.03
N	2.3	0.08	0.6	0.03
Total	30.4		22.7	

This adjusted composition is used to define the secondary stream (volatile) composition.

The lower heating value of the volatiles can be computed from the known heating value of the coal and the char (DAF):

- LCV $_{coal,DAF}$ = 35.3 MJ/kg
- LCV $_{char,DAF}$ = 32.9 MJ/kg

You can compute the heating value of the volatiles as

$$\mathrm{LCV}_{vol}=\frac{35.3MJ/kg-0.696\times 32.9MJ/kg}{0.304} \tag{17–2}$$

or

$$\mathrm{LCV}_{vol}=40.795MJ/kg \tag{17–3}$$

Note that for both of these composition input methods, you should take care to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should only be selected if you are using the empirical fuel input method.

17.4.5.3. Additional Coal Modeling Inputs in ANSYS FLUENT

Within ANSYS FLUENT, the DPM coal combustion simulation is defined as usual when the non-premixed combustion model is selected. The air (oxidizer) inlets are defined as having a mixture fraction value of zero. No gas phase fuel inlets will be included and the sole source of fuel will come from the coal devolatilization and char burnout. The coal particles are defined as injections using the **Set Injection Properties** dialog box in the usual way, and physical properties for the coal material are specified as described in *Setting Material*

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties for the Discrete Phase (p. 1054). You should keep in mind the following issues when defining injections and discrete-phase material properties for coal materials:

- In the **Set Injection Properties** dialog box, you will specify for the **Oxidizing Species** one of the components of the oxidizer stream. This species concentration field will be used to calculate the diffusion-controlled driving force in the char burnout law (if applicable), and is O_2 by default.

  The specification of the char and volatile streams differs depending on the type of model you are defining:

  - If coal is modeled using a single mixture fraction, the gas phase species representing the volatiles and the char combustion are represented by the mixture fraction used by the non-premixed combustion model.
  - If coal is modeled using two mixture fractions, rather than specifying a destination species for the volatiles and char, you will instead specify the **Devolatilizing Stream** (as secondary) and the **Char Stream** (as primary).
  - If coal is modeled using one mixture fraction, and another fuel is modeled using a second mixture fraction, you should specify the stream representing the coal as both the **Devolatilizing Stream** and the **Char Stream**.
- In the **Create/Edit Materials** dialog box, **Vaporization Temperature** should be set equal to the fuel inlet temperature. This temperature controls the onset of the devolatilization process. The fuel inlet temperature that you define in the **Boundary** tab of the **Species Model** dialog box should be set to the temperature at which you want to initiate devolatilization. This way, the look-up tables will include the appropriate temperature range for your process.
- In the **Create/Edit Materials** dialog box, **Volatile Component Fraction** and **Combustible Fraction** should be set to values that are consistent with the coal composition used to define the fuel (and secondary) stream composition.
- Also in the **Create/Edit Materials** dialog box, you will be prompted for the **Burnout Stoichiometric Ratio** and for the **Latent Heat**. The **Burnout Stoichiometric Ratio** is used in the calculation of the diffusion controlled burnout rate but has no other impact on the system chemistry when the non-premixed combustion model is used. The **Burnout Stoichiometric Ratio** is the mass of oxidant required per mass of char. The default value of 1.33 assumes that C(s) is oxidized by O_2 to yield CO. The **Latent Heat** input determines the heat required to generate the vapor phase volatiles defined in the non-premixed system chemistry. You can usually set this value to zero when the non-premixed model is used, since your definition of volatile species will have been based on the overall heating value of the coal. However, if the coal composition includes the water content, the latent heat should be set as follows:
  - Set the latent heat to zero if the water content of the coal has been defined as **H2O(L)**. In this case, the system chemistry will include the latent heat required to vaporize the liquid water.
  - Set the latent heat to the value for water (2.25×10^6 J/kg), adjusted by the mass loading of water in the volatiles, if the water content of the coal has been defined using water vapor, **H2O**. In this case, the water content you defined will be evolved along with the other species in the coal but the system chemistry does not include the latent heat effect.
- The **Density** you define for the coal in the **Create/Edit Materials** dialog box should be the apparent density, including ash content.
- You will not be asked to define the **Heat of Reaction for Burnout** for the char combustion.

17.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion

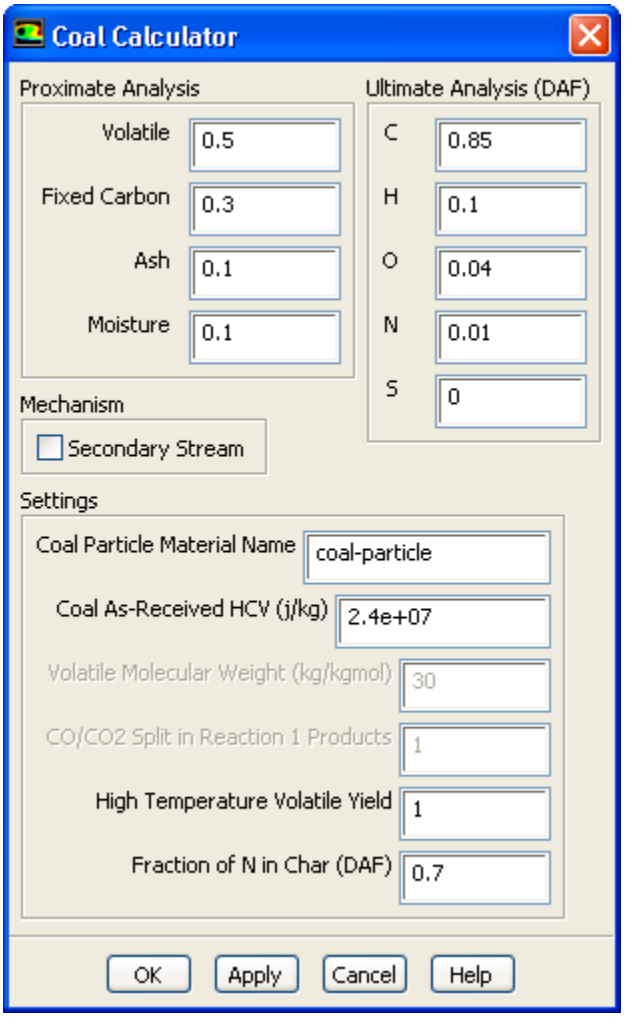
ANSYS FLUENT reports the rate of volatile release from the coal using the **DPM Evaporation/Devolatilization** postprocessing variable. The rate of char burnout is reported in the **DPM Burnout** variable.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.4.5.5. The Coal Calculator

The **Coal Calculator** dialog box automates the calculations described above for setting up a coal case from the proximate and ultimate analyses.

Figure 17.11 The Coal Calculator Dialog Box



The inputs to the **Coal Calculator** dialog box are:

1. Coal **Proximate Analysis**, which is the mass fraction of **Volatile, Fixed Carbon, Ash** and **Moisture** in the coal.
2. Coal **Ultimate Analysis**, which is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal.
3. The option to use a **Secondary Stream**. If enabled, the two mixture fraction model will be set with the primary stream representing char as $C<s>$, and an empirical secondary stream representing the volatiles.
4. The **Coal Particle Material Name**. A DPM Combusting Particle Material will be created with this name. The default name is *coal-particle*.
5. The **Coal As-Received HCV** (Higher Calorific Value).
6. The **High Temperature Volatile Yield**. Enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction. To model this, the actual volatile fraction used

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is calculated as that specified in the **Proximate Analysis** input multiplied by the **High Temperature Volatile Yield**. The actual **Fixed Carbon** fraction is then calculated as one minus the sum of the actual **Volatile**, **Ash**, and **Moisture** fractions.

7. **Fraction of N in Char (DAF)**. This input is used in calculating the split of atomic nitrogen for the Fuel NOx model.

When the **OK** button is clicked, ANSYS FLUENT makes the following changes:

a. The empirical fuel atomic compositions in the **Boundary** tab are set, and the **Non-Adiabatic** model is enabled as required for DPM. The empirical fuel (DAF volatile) Lower Calorific Value (LCV_{vol}) is calculated as follows. First the DAF LCV of the coal is computed as,

$$LCV_{coal}^{DAF} = \frac{HCV_{coal}^{ar} - h_{H_2O}^{latent} Moisture}{1 - Moisture - Ash} - \frac{H_{ar} W_{H_2O}}{2 W_H} h_{H_2O}^{latent} \quad \textbf{(17–4)}$$

where $Moisture$ and Ash are the proximate moisture and ash fractions, H_{ar} is the ultimate H fraction, W_{H_2O} and W_H are the molecular weight of water and atomic hydrogen, respectively, and $h_{H_2O}^{latent}$ is the latent heat of water.

LCV_{vol} is calculated from LCV_{coal}^{DAF} using,

$$LCV_{vol} = \frac{LCV_{coal}^{DAF}(1 - Moisture - Ash) - LCV_{char} FixedCarbon}{Volatile} \quad \textbf{(17–5)}$$

where $FixedCarbon$ and $Volatile$ are the proximate fixed carbon and volatile fractions, respectively.

b. A combusting particle material is created with **Volatile Component Fraction** and **Combustible Fraction** calculated from the ultimate and proximate analyses. The Discrete Phase Model (DPM) is enabled.

c. For the Fuel NOx model, the char N conversion is set to *NO*, and the Fuel NOx **Volatile** and **Char** mass fractions are set according to the ultimate and proximate compositions. Note that even though some of the Fuel NOx parameters are changed, the Fuel NOx model itself is not enabled.

After the **Coal Calculator** has set up the relevant models, you must build the PDF Table by clicking **Calculate PDF Table** in the **Table** tab. You will also need to create injections if you have not done this yet. After converging your coal combustion case, you may want to enable the NOx model for post-processing nitrogen-oxide pollutants.

17.5. Setting Up Control Parameters

For information about setting up control parameters, please see the following sections:

17.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species
17.5.2. Defining the Flamelet Controls
17.5.3. Zeroing Species in the Initial Unsteady Flamelet

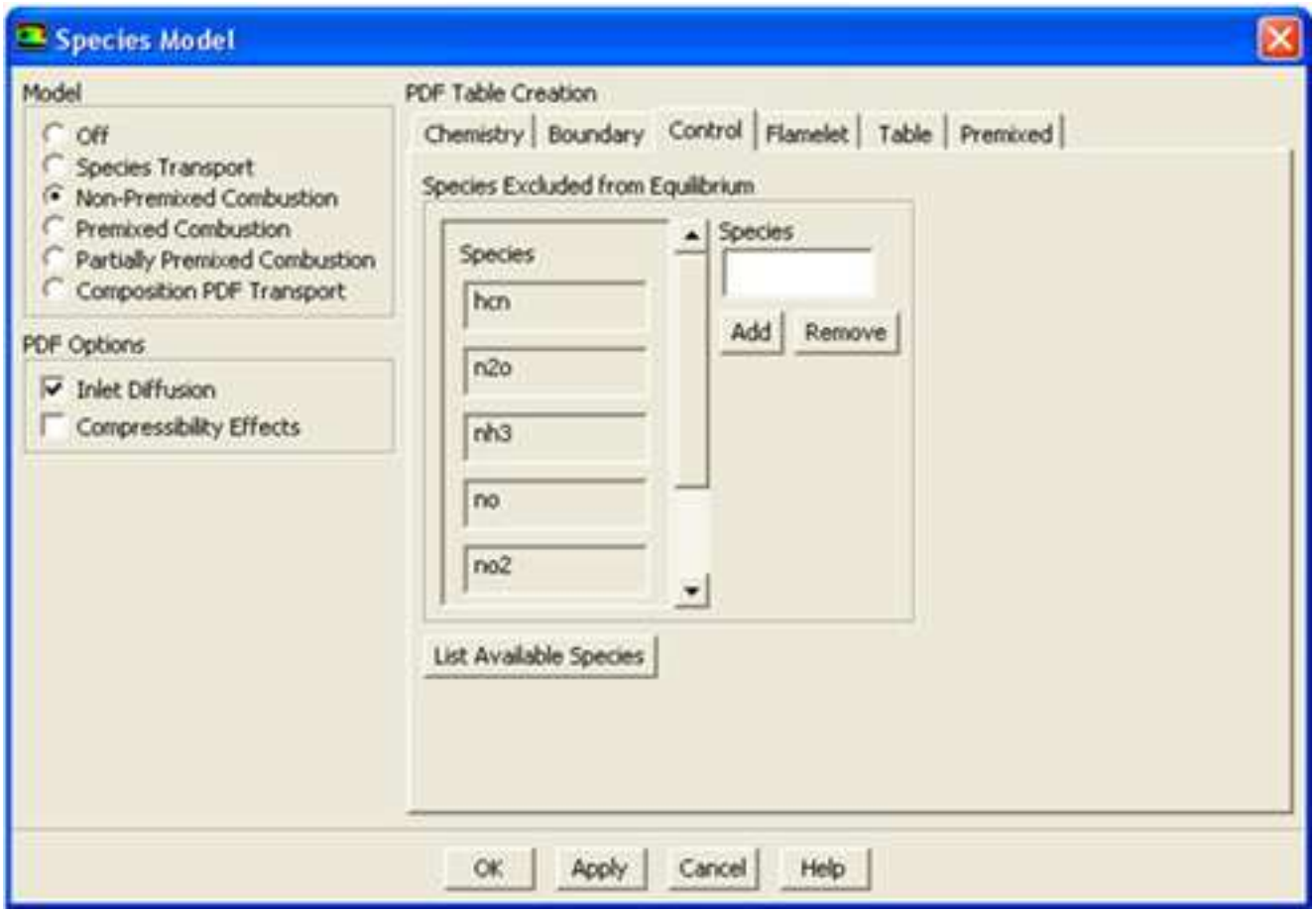
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species

Because ANSYS FLUENT calculates all intermediate and product species automatically during the equilibrium calculation, certain species will be included that are generally not in chemical equilibrium. Principal among these are the NOx species. Specifically, the NOx reaction rates are slow and should not be treated using an equilibrium assumption. Instead, the NOx concentration is predicted most accurately using the ANSYS FLUENT NOx postprocessor, where finite-rate kinetics are included (see *NOx Formation* (p. 943)). The NOx species can be safely excluded from the equilibrium calculation since they are present at low concentrations and have little impact on the density, temperature, and other species.

To force the exclusion of a species from the equilibrium calculation, click the **Control** tab in the **Species Model** dialog box (*Figure 17.12* (p. 862)).

Figure 17.12 The Species Model Dialog Box (Control Tab)



Under **Species Excluded From Equilibrium**, enter the chemical formula for the desired species in the **Add/Remove Species** field. Next, click **Add** to add the species to the **Species** list or **Remove** to remove an existing species from the **Species** list.

If there are species that you want to include in your PDF table that would be ignored by ANSYS FLUENT due to their low concentration (e.g., CH, CH_2, CH_3 for the NOx calculation), you can force ANSYS FLUENT to include them using the text interface:

`define` → `models` → `species` → `non-premixed-combustion`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

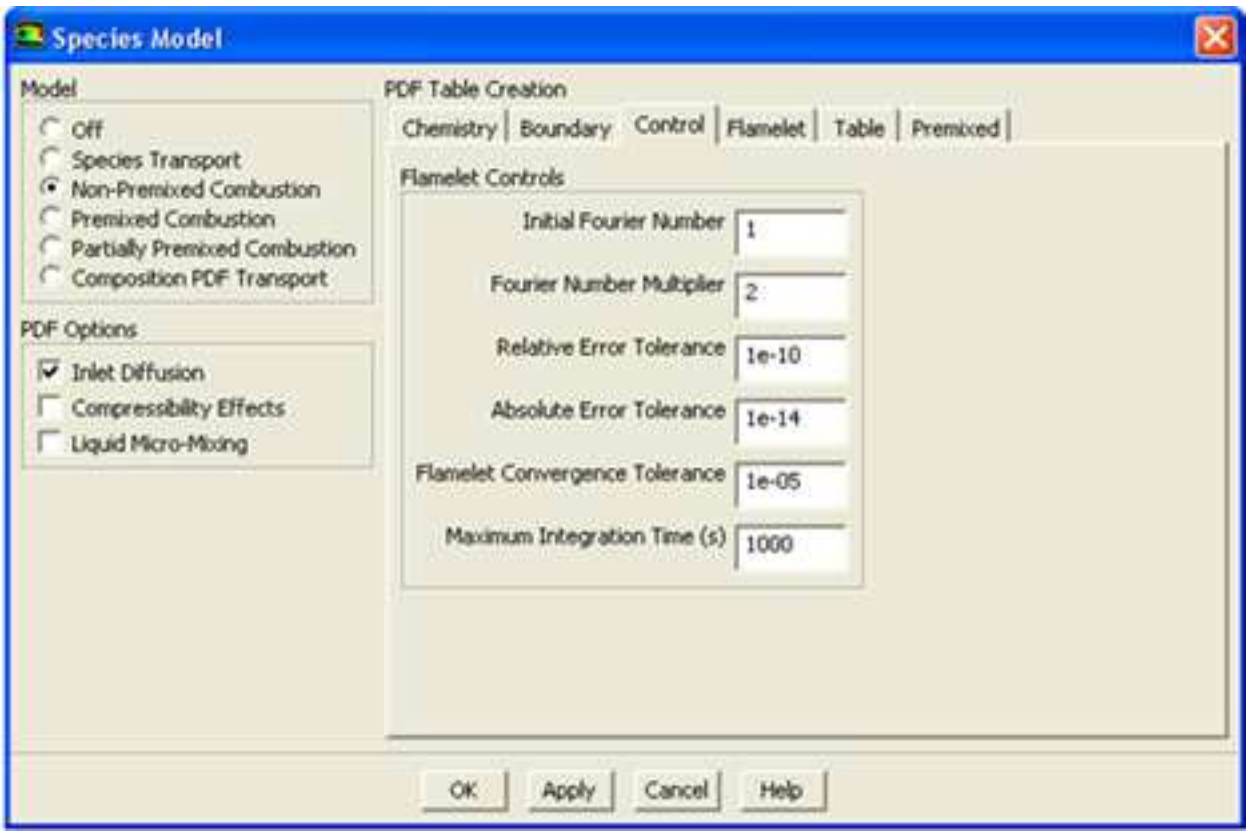
When the console window prompts you with `Force Equilibrium Species to Include...`, specify the appropriate species by entering the chemical formula(s) in double quotes (e.g., `"ch"`, `"ch2"`).

Note that you will have to first set up the inputs for the fuel and oxidizer before you are given the option to include the species.

17.5.2. Defining the Flamelet Controls

When the steady laminar flamelet model is selected, and you have created or imported a flamelet, you can adjust the controls for the flamelet solution in the **Control** tab of the **Species Model** dialog box (*Figure 17.13* (p. 863)).

Figure 17.13 The Species Model Dialog Box (Control Tab) for the Steady Laminar Flamelet Model



The **Initial Fourier Number** sets the first time step for the solution of the flamelet equations (Equation 8–43 and Equation 8–44 in the Theory Guide). This first time step is calculated as the explicit stability-limited diffusion time step multiplied by this value. If the solution diverges before the first time step is complete, the value should be lowered.

The **Fourier Number Multiplier** increases the time step at subsequent times. Every time step after the first is multiplied by this value. If the solution diverges after the first time step, this value should be reduced.

During the numerical integration of the flamelet equations, the local error is controlled to be less than

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$error_{loc,i} = \varepsilon_{rel}\phi_i + \varepsilon_{abs} \tag{17–6}$$

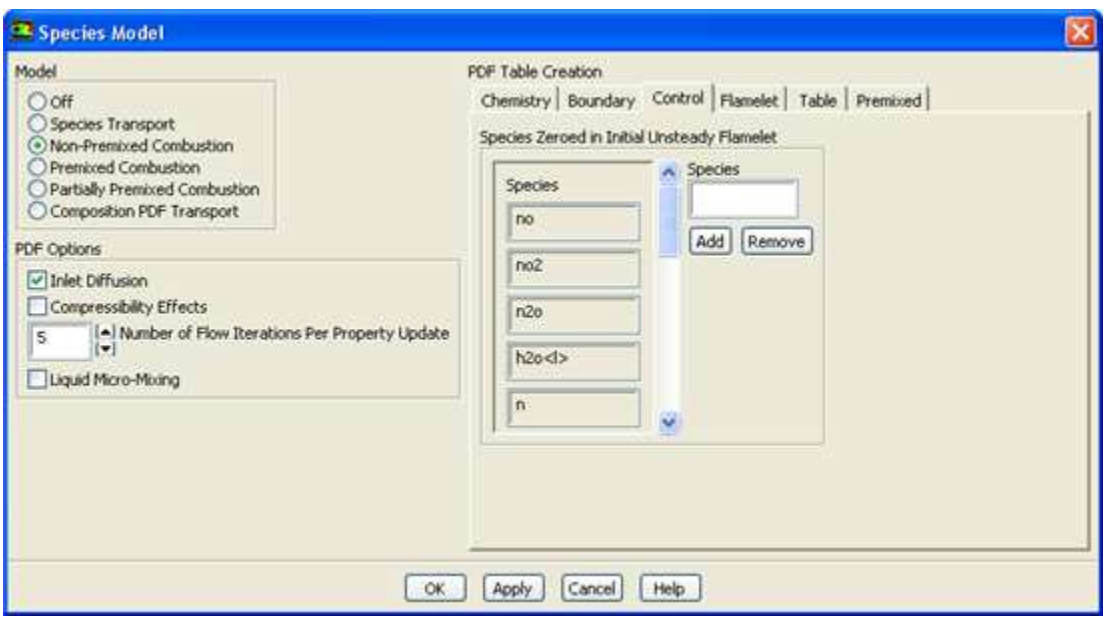
where ϕ_i represents the species mass fractions and temperature at point i in the 1D flamelet. ε_{rel} is the value of the **Relative Error Tolerance** and ε_{abs} is the value of the **Absolute Error Tolerance**, both of which you can specify.

Because steady laminar flamelets are obtained by time-stepping, they are considered converged only when the maximum absolute change in species fraction or temperature at any discrete mixture-fraction point is less than the specified **Flamelet Convergence Tolerance**. Between time steps, the flamelet species fractions and temperature will sometimes oscillate, which causes absolute changes that are always greater than the flamelet convergence tolerance. In such cases, ANSYS FLUENT will stop the flamelet calculation after the total elapsed time has exceeded the **Maximum Integration Time**.

17.5.3. Zeroing Species in the Initial Unsteady Flamelet

When modeling gas-phase combustion using the Eulerian unsteady laminar flamelet model, the flamelet fields are initialized to a burning, steady-flamelet solution in order to model ignition. However, assuming steady-flamelet profiles for slow-forming species is inaccurate. A better approximation is to identify the slow species and to set them to zero, which is done in the **Control** tab. By default, ANSYS FLUENT selects some NOx species (NO, NO_2, N_2O, N, NH, NH_2, NH_3, NNH, HCN, HNO, CN, $H2CN$, $HCNN$, $HCNO$, $HOCN$, $HNCO$, HCO), as well as liquid water $H_2O<l>$ and solid carbon $C<s>$ to be zeroed. See *Figure 17.14* (p. 864).

Figure 17.14 Method to Zero Out the Slow Chemistry Species



17.6. Calculating the Flamelets

For information about calculating flamelets, please see the following sections:

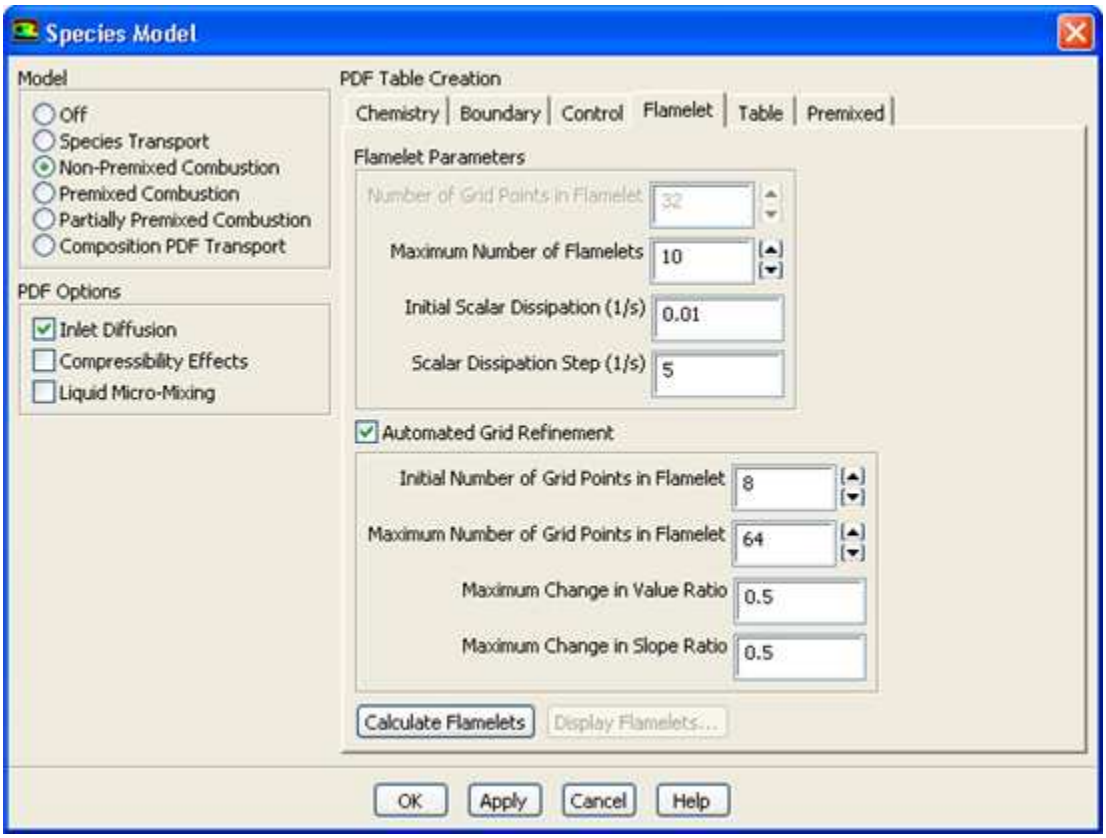
17.6.1. Steady Flamelet
17.6.2. Unsteady Flamelet
17.6.3. Saving the Flamelet Data
17.6.4. Postprocessing the Flamelet Data

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.6.1. Steady Flamelet

In the **Flamelet** tab of the **Species Model** dialog box (*Figure 17.15* (p. 865)), you will enter values for parameters of the flamelet(s).

Figure 17.15 The Species Model Dialog Box (Flamelet Tab)



The **Flamelet Parameters** are as follows:

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since the flamelet species and temperature are solved coupled and implicit in f space, the solution time and memory requirements increase greatly with the number of f grid points.

Maximum Number of Flamelets

specifies the maximum number of laminar flamelet profiles to be calculated. If the flamelet extinguishes before this number is reached, flamelet generation is halted and the actual number of flamelets in the flamelet library will be less than this value.

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library. This corresponds to χ_0 in Equation 8–49 in the Theory Guide.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated. This corresponds to $\Delta\chi$ in Equation 8–49 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automated Grid Refinement employs an adaptive algorithm, which inserts grid points so that the change of values, as well as the change of slopes, between successive grid points is less than user specified tolerances. For information about this option, refer to Steady Laminar Flamelet Automated Grid Refinement in the Theory Guide.

Initial Number of Grid Points in Flamelet
calculates a steady solution on a coarse grid, with a default of 8. See Equation 8–50 in the Theory Guide.

Maximum Number of Grid Points in Flamelet
has a default of 64.

Maximum Change in Value Ratio
has a default of 0.5 and is ε_v in Equation 8–50 in the Theory Guide.

Maximum Change in Slope Ratio
has a default of 0.5 and is ε_s in Equation 8–50 in the Theory Guide.

Click **Calculate Flamelets** to begin the laminar flamelet calculation. Sample output for a flamelet calculation is shown below.

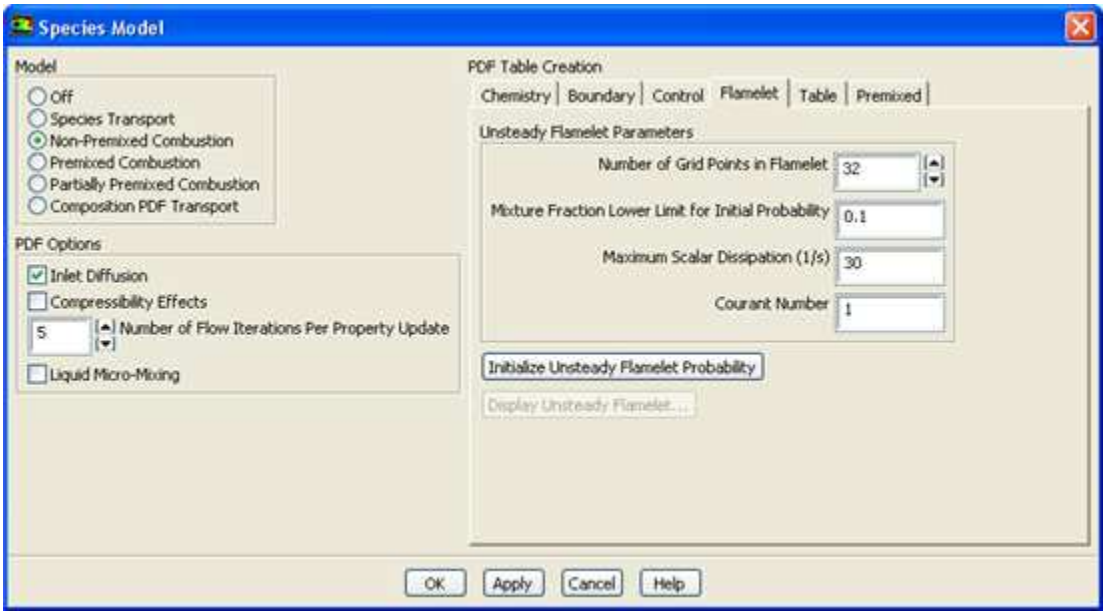
```
 Generating flamelet 1 at scalar dissipation  0.01 /s

Time (s)   Temp (K)   Residual
1.679e-05   2233.7   3.779e+00
5.038e-05   2233.0   7.734e-02
1.175e-04   2231.5   1.648e-01
2.519e-04   2228.6   3.652e-01
5.206e-04   2223.6   8.295e-01
1.058e-03   2215.7   2.100e+00
2.133e-03   2205.5   3.540e+00
4.282e-03   2197.0   4.607e+00
8.581e-03   2193.6   6.639e+00
1.718e-02   2193.1   4.905e+00
3.437e-02   2193.4   5.792e+00
6.877e-02   2194.3   4.659e+00
1.375e-01   2195.3   3.922e+00
2.751e-01   2192.2   3.181e+00
5.502e-01   2188.6   2.549e+00
1.100e+00   2184.8   1.639e+00
2.201e+00   2182.9   4.604e+00
4.402e+00   2186.8   1.307e+00
8.804e+00   2189.6   4.420e-01
1.761e+01   2190.0   8.581e-02
3.522e+01   2190.0   1.199e-02
7.043e+01   2190.0   1.735e-03
1.409e+02   2190.0   4.217e-04
2.817e+02   190.0    6.892e-05
5.635e+02   2190.0   6.777e-06

Flamelet successfully generated
```

17.6.2. Unsteady Flamelet

In the **Flamelet** tab of the **Species Model** dialog box (*Figure 17.16* (p. 867)), you will enter values for parameters of the flamelet.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.16 The Flamelet Tab for the Unsteady Laminar Flamelet Model



The **Unsteady Flamelet Parameters** are as follows:

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since the flamelet species and temperature are solved coupled and implicit in f space, the solution time and memory requirements increase with the number of f grid points.

Mixture Fraction Lower Limit for Initial Probability

is the mixture fraction above which the marker probability will be initialized to 1, and below which the marker probability will be initialized to 0. In general, it should be set greater than the stoichiometric mixture fraction.

Maximum Scalar Dissipation

is where flamelets extinguish at large scalar dissipation (mixing) rates. To prevent excessive mixing in the flamelet, ANSYS FLUENT allows you to specify a **Maximum Scalar Dissipation** rate for the 1D flamelet equations. A reasonable value for this is the steady flamelet extinction scalar dissipation. The default value of 30/s is near the steady extinction scalar dissipation of a methane-air flame at standard temperature and pressure.

Courant Number

is the number at which ANSYS FLUENT automatically selects the time step for the probability equation based on this convective Courant number.

Click **Initialize Unsteady Flamelet Probability** to initialize the unsteady flamelet and its probability marker equation. ANSYS FLUENT is now ready for postprocessing the 1D unsteady flamelet and the 2D/3D unsteady marker probability equation.

17.6.3. Saving the Flamelet Data

The flamelet tables may be written to file for import into later sessions of ANSYS FLUENT. You may want to do this, for example, to change the number of discretization points in the PDF table, or to plot the flamelet profiles in ANSYS FLUENT. The flamelet tables should be saved before you create the PDF table:

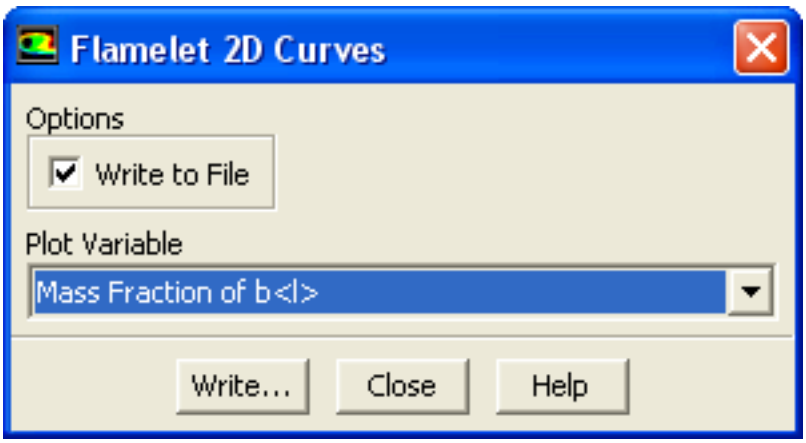
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → **Write** → **Flamelet...**

17.6.4. Postprocessing the Flamelet Data

For the flamelet model, you can display or write flamelet curves. Click the **Display Flamelets...** or **Display Unsteady Flamelet...** button. If you have a single flamelet, as for the unsteady flamelet model, you can access the **Flamelet 2D Curves** dialog box (*Figure 17.17* (p. 868)).

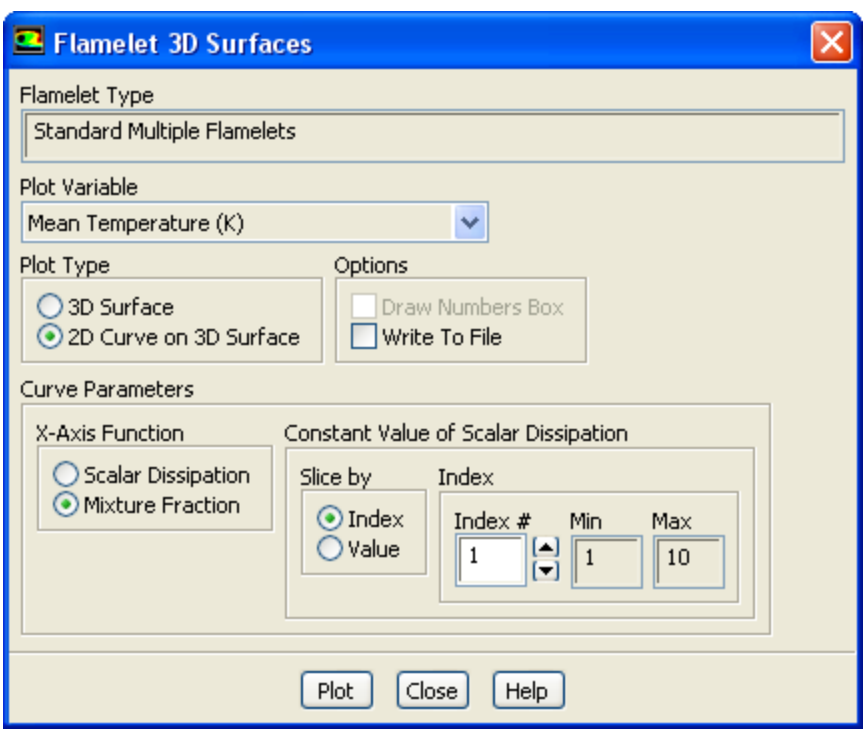
Figure 17.17 The Flamelet 2D curves Dialog Box



For the steady flamelet model with more than one flamelet, you can display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation using the **Flamelet 3D Surfaces** dialog box (e.g., *Figure 17.18* (p. 868)).

To access this dialog box, click the **Display Flamelets...** button in the **Flamelet** tab of the **Species Model** dialog box, as shown in *Figure 17.15* (p. 865).

Figure 17.18 The Flamelet 3D Surfaces Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To display the flamelet tables graphically, use the following procedure:

1. In the **Flamelet 3D Surfaces**
2. Specify the **Plot Type** as either **3D Surface** or **2D Curve on 3D Surface**.
   - For a 3D surface, enable or disable **Draw Numbers Box** under **Options**. When this option is turned on, the display will include a wireframe box with the numerical limits in each coordinate direction.
   - For a 2D curve on a 3D surface:
     a. Specify whether you want to write the plot data to a file by toggling **Write To File** under **Options**.
     b. Specify the **X-Axis Function** against which the plot variable will be displayed by selecting **Scalar Dissipation** (χ), or **Mixture Fraction** (f). The variable that is not selected will be held constant.
     c. Specify the type of discretization (i.e., how the flamelet data will be sliced) for the variable that is being held constant (under **Constant Value of Mixture Fraction** or **Constant Value of Scalar Dissipation**).
        – If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Flamelet** tab of the **Species Model** dialog box (see *Calculating the Flamelets* (p. 864)).
        – If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.
3. Write or display the flamelet table results. If you have turned on the **Write To File** option for a 2D plot, click **Write** and specify a name for the file in *The **Select File** Dialog Box* (p. 33). Otherwise, click **Plot** or **Display** as appropriate to display a 2D plot or 3D surface in the graphics window.

Figure 17.19 (p. 870) and *Figure 17.20* (p. 870) show examples of a 2D curve plot and 3D surface plot of a flamelet table.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.19 Example 2D Plot of Flamelet Data

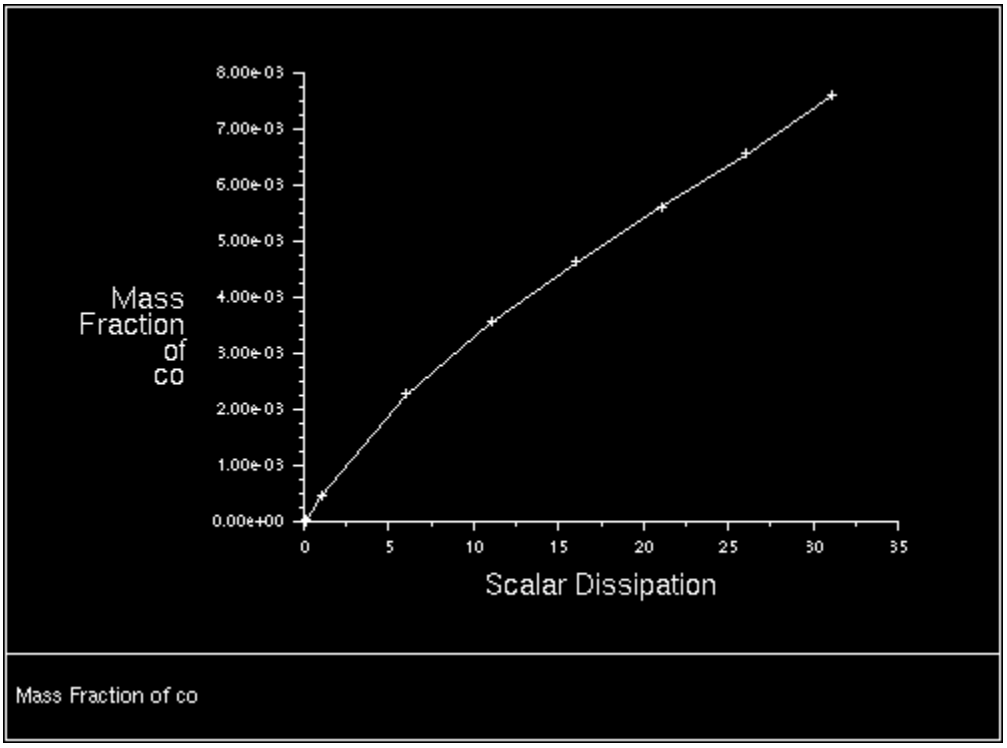
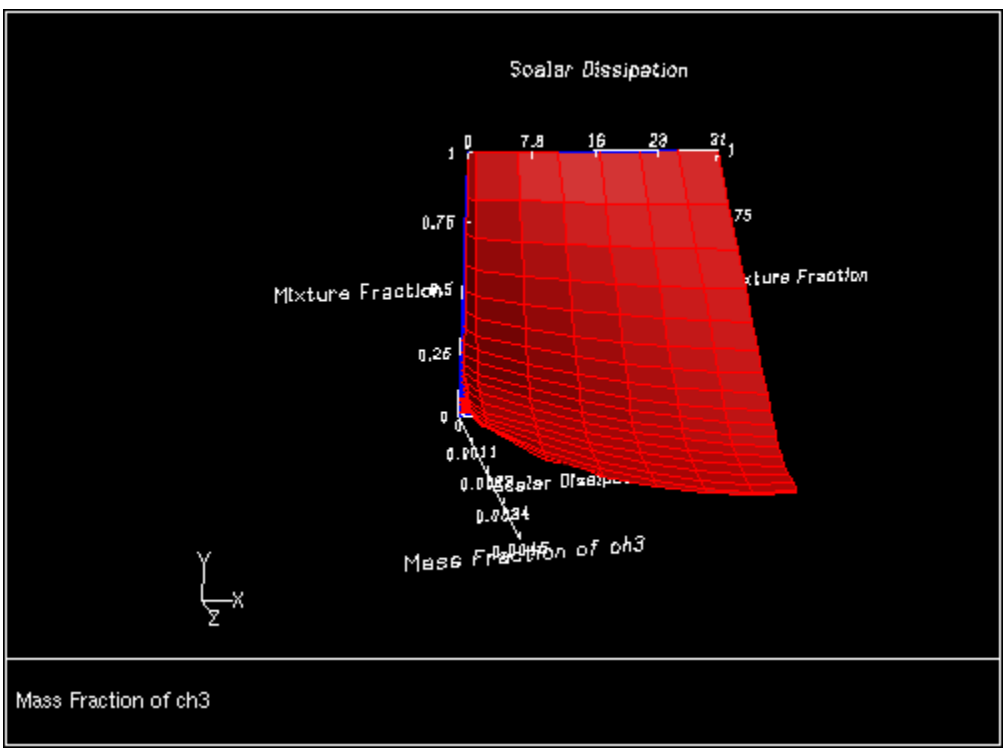


Figure 17.20 Example 3D Plot of Flamelet Data

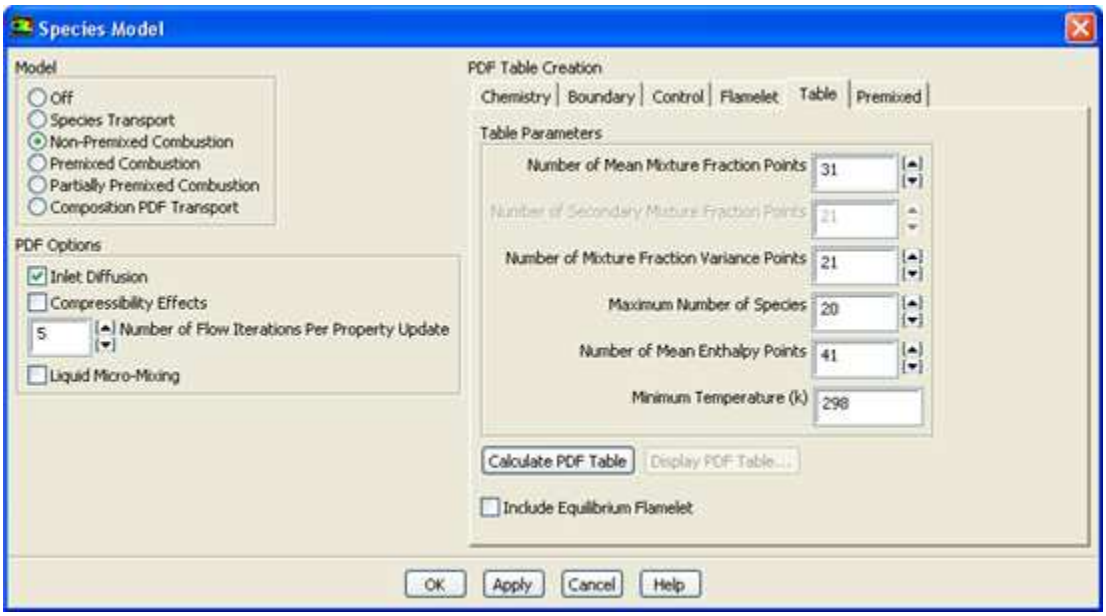


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.7. Calculating the Look-Up Tables

ANSYS FLUENT requires additional inputs that are used in the creation of the look-up tables. Several of these inputs control the number of discrete values for which the look-up tables will be computed. These parameters are input in the **Table** tab of the **Species Model** dialog box (e.g., *Figure 17.21* (p. 871)).

Figure 17.21 The Species Model Dialog Box (Table) Tab



The look-up table parameters are as follows:

Number of Mean Mixture Fraction Points

is the number of discrete values of $\overline{f}$ at which the look-up tables will be computed. For a two-mixture-fraction model, this value is the number of points in the instantaneous state profile used to compute the PDF if you choose the β PDF model (see *Tuning the PDF Parameters for Two-Mixture-Fraction Calculations* (p. 888)). Increasing the number of points will yield a more accurate PDF shape, but the calculation will take longer. The mean mixture fraction points will be automatically clustered around the stoichiometric mixture fraction value.

Number of Mixture Fraction Variance Points

is the number of discrete values of $\overline{f'^2}$ at which the look-up tables will be computed. Lower resolution is acceptable because the variation along the $\overline{f'^2}$ axis is, in general, slower than the variation along the $\overline{f}$ axis of the look-up tables. This option is available only when no secondary stream has been defined.

Number of Secondary Mixture Fraction Points

is the number of discrete values of p_{sec} at which the look-up tables will be computed. Like the **Number of Mean Mixture Fraction Points**, ANSYS FLUENT will use the **Number of Secondary Mixture Fraction Points** to compute the equilibrium state-relation if you choose the β PDF option (see *Tuning the PDF Parameters for Two-Mixture-Fraction Calculations* (p. 888)) for a two-mixture-fraction model. A larger number of points will give a more accurate shape for the PDF, but with a longer calculation time. This option is available only when a secondary stream has been defined.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables. The maximum number of species that can be included is 100. Note that the maximum number of species for the equilibrium computations is 500, and the maximum number of species for the flamelet generation and importing is 300. ANSYS FLUENT will automatically select the species with the largest mole fractions to include in the PDF table.

Number of Mean Enthalpy Points

is the number of discrete values of enthalpy at which the look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system. The number of points required will depend on the chemical system that you are considering, with more points required in high heat release systems (e.g., hydrogen/oxygen flames). This option is not available with the unsteady flamelet model.

Minimum Temperature

is used to determine the lowest temperature for which the look-up tables are generated (see Figure 8.10: "Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems" in the Theory Guide). Your input should correspond to the minimum temperature expected in the domain (e.g., an inlet or wall temperature). The minimum temperature should be set 10–20 K below the minimum system temperature. This option is available only if you are modeling a non-adiabatic system. This option is not available with the unsteady flamelet model.

Include Equilibrium Flamelet

specifies that an equilibrium flamelet (i.e., $\chi = 0$) will be generated in ANSYS FLUENT and appended to the flamelet library before the PDF table is calculated. This option is available when generating or importing multiple flamelets, as well as when a single flamelet is considered. In the latter case, the PDF table will consist of two scalar dissipation slices, namely the equilibrium slice at $\chi = 0$, and the flamelet slice. This option is not available with the equilibrium chemistry model or the unsteady flamelet model.

When you are satisfied with your inputs, click **Calculate PDF Table** to generate the look-up tables.

The computations performed for a single-mixture-fraction calculation culminate in the discrete integration of Equation 8–16 (or Equation 8–24 in the Theory Guide) as represented in Figure 8.5: "Logical Dependence of Averaged Scalars on Mean Mixture Fraction, the Mixture Fraction Variance, and the Chemistry Model (Adiabatic, Single-Mixture-Fraction Systems)" (or Figure 8.6: "Logical Dependence of Averaged Scalars on Mean Mixture Fraction, the Mixture Fraction Variance, Mean Enthalpy, and the Chemistry Model (Non-Adiabatic, Single-Mixture-Fraction Systems)" in the Theory Guide). For a two-mixture-fraction calculation, ANSYS FLUENT will calculate the physical properties using Equation 8–14 or its adiabatic equivalent. The computation time will be shortest for adiabatic single-mixture-fraction equilibrium calculations and longest for non-adiabatic calculations involving multiple flamelet generation. Below, sample outputs are shown for an adiabatic single-mixture-fraction equilibrium calculation and a non-adiabatic calculation with laminar flamelets:

```
Generating PDF lookup table
 Type of the PDF Table: Adiabatic Table (Two Streams)
Calculating table .....

1271 points calculated
22 species added
PDF Table successfully generated!

Generating PDF lookup table
 Type of the PDF Table: Nonadiabatic Table with Strained Flamelet Model (Two Streams)
 Calculating table .....
 calculating temperature limits .....
 calculating temperature limits .....
 calculating scalar dissipation slices .....
  - scalar dissipation slice  9
 calculating equilibrium slice .....
Performing PDF integrations.....
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  16810 points calculated
  17 species added
  PDF Table successfully generated!
 Initializing PDF table arrays and structures.
```

Important

Note that there is a significant difference in run-time between the one-mixture fraction model and the two-mixture fraction model. In the one-mixture fraction model, the PDF table contains the mean data of density, temperature, and specific heats, and is three-dimensional for an equilibrium nonadiabatic case (mean mixture fraction, mixture fraction variance, and mean heat loss). For this case, ANSYS FLUENT updates properties every flow iteration. In the case of the two-mixture fraction model, only the instantaneous state relationships are stored and mean properties are calculated from these by performing PDF integrations in every cell of the ANSYS FLUENT simulation. Since this is computationally expensive, ANSYS FLUENT provides the option of only updating properties after a specified number of iterations.

After completing the calculation at the specified number of mixture fraction points, ANSYS FLUENT reports that the calculation succeeded. In a single-mixture-fraction case, the resulting look-up tables take the form illustrated in Figure 8.8: "Visual Representation of a Look-Up Table for the Scalar (Mean Value of Mass Fractions, Density, or Temperature) as a Function of Mean Mixture Fraction and Mixture Fraction Variance in Adiabatic Single-Mixture-Fraction Systems" in the Theory Guide (or Figure 8.10: "Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems", for non-adiabatic systems). These look-up tables can be plotted using the available graphics tools, as described in *Postprocessing the Look-Up Table Data* (p. 874).

Note that in non-adiabatic calculations, the console window will report that the temperature limits and enthalpy slices have been calculated.

For a two-mixture-fraction case, the resulting look-up tables take the form illustrated in Figure 8.9: "Visual Representation of a Look-Up Table for the Scalar ϕ_I as a Function of Fuel Mixture Fraction and Secondary Partial Fraction in Adiabatic Two-Mixture-Fraction Systems" in the Theory Guide (or Figure 8.11: "Visual Representation of a Look-Up Table for the Scalar ϕ_I as a Function of Fuel Mixture Fraction and Secondary Partial Fraction, and Normalized Heat Loss/Gain in Non-Adiabatic Two-Mixture-Fraction Systems", for non-adiabatic systems).

17.7.1. Full Tabulation of the Two-Mixture-Fraction Model

The default algorithm for the two-mixture-fraction model is to perform PDF integrations of the equilibrium state relationships at run-time. Since these are multi-dimensional integrals, the default two-mixture-fraction model can be computationally demanding.

Alternatively, you may want to pre-compute these integrations and create 4D look-up tables for adiabatic simulations, or 5D tables for non-adiabatic simulations. Such high-dimensional tables are computationally expensive to build, and may require large memory and disk storage, but can offer substantial improvement in run-time speed.

The option to create a fully-tabulated two-mixture-fraction table is available in cases with the two-mixture-fraction model enabled, via the text command:

`define/models/species/full-tabulation?`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are modeling pollutant formation, please note that the full tabulation option is not compatible with the **mixture-fraction** option for **PDF Mode** in the **Turbulence Interaction Mode** tab settings. See *Setting Turbulence Parameters* (p. 955) for details on Turbulence Interaction options for pollutant modeling.

17.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables

Complex chemistry and non-adiabatic effects may make the equilibrium calculation more time-consuming and difficult. In some instances the equilibrium calculation may even fail. You may be able to eliminate any difficulties that you encounter by trying the calculation as an adiabatic system. Adiabatic system calculations are generally quite straightforward and can provide valuable insight into the optimal inputs to the non-adiabatic calculation.

Additional stability issues may arise for solid or heavy liquid fuels that have been defined using the empirical fuel approach. You may find that, for rich fuel mixtures, the equilibrium calculation produces very low temperatures and eventually fails. This indicates that strong endothermic reactions are taking place and the mixture is not able to sustain them. In this situation, you may need to raise the heating value of the fuel until ANSYS FLUENT produces acceptable results. Provided that your fuel will be treated as a liquid or solid (coal) fuel, you can maintain the desired heating value in your ANSYS FLUENT simulation. This is accomplished by defining the difference between the desired and the adjusted heating values as latent heat (in the case of combusting solid fuel) or heat of pyrolysis (in the case of liquid fuel).

17.7.3. Saving the Look-Up Tables

The look-up tables may be stored in a file that you can read back into later sessions of ANSYS FLUENT. The look-up tables should be saved before you exit from the current ANSYS FLUENT session.

File → **Write** → **PDF...**

By default, the file will be saved as formatted (ASCII, or text). To save a binary (unformatted) file, turn on the **Write Binary Files** option in *The **Select File** Dialog Box* (p. 33).

17.7.4. Postprocessing the Look-Up Table Data

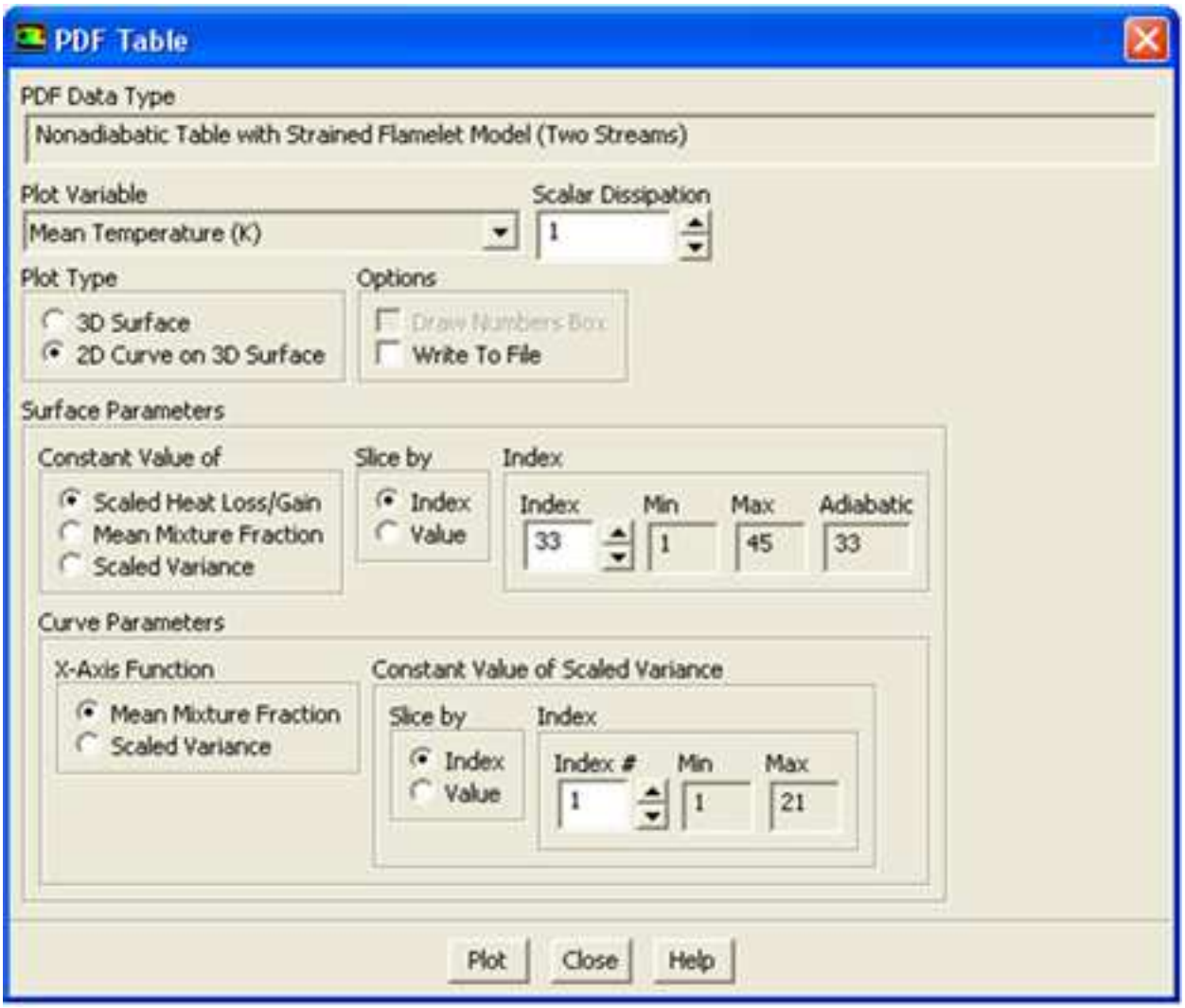
It is important for you to view your temperature and species tables to ensure that they are adequately but not excessively resolved. Inadequate resolution will lead to inaccuracies, and excessive resolution will lead to unnecessarily slow calculation times.

After a PDF table has been generated or read into ANSYS FLUENT, you can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy using the **PDF Table** dialog box (e.g., *Figure 17.22* (p. 875)). The **PDF Table** dialog box can be accessed in one of two ways: you can click on the **Display PDF Table...** button in the **Table** tab of the **Species Model** dialog box (as shown in *Figure 17.21* (p. 871)) or you can use the path

Display → **PDF Tables/Curves...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.22 The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets)



To display the look-up tables graphically, use the following procedure:

1. In the **PDF Table** dialog box, in the **Plot Variable** drop-down list you can select temperature, density, or species fraction as the variable to be plotted.
2. (multiple flamelets only) Specify the value of the **Scalar Dissipation**. In the case of non-adiabatic flamelets, there is the additional parameter of mean enthalpy. In addition to varying the mean enthalpy and mean mixture fraction, you can vary the display of the PDF table by changing the value of the scalar dissipation, which gives the table a fourth "dimension".
3. Specify the **Plot Type** as either **3D Surface** or **2D Curve on 3D Surface**. In the equilibrium model, a 2D curve is a slice of a 3D surface, and thus some options selected for a 3D surface may impact the display of a 2D curve.
   - For a 3D surface:
     a. Enable or disable **Draw Numbers Box** under **Options**. When this option is turned on, the display will include a wireframe box with the numerical limits in each coordinate direction.
     b. (non-adiabatic cases only) Under **Surface Parameters**, specify the discrete independent variable to be held constant in the look-up table (**Constant Value of**).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For a single-mixture-fraction case, select **Scaled Heat Loss/Gain** ($\overline{H}$), **Mean Mixture Fraction** ($\overline{f}$), or **Scaled Variance** ($\overline{f'^2}$). For any mean mixture fraction $\overline{f}$, the variance varies between a minimum of 0 and a maximum of $\overline{f}(1-\overline{f})$. In order to view the mixture fraction variance, it is normalized by *Equation 17–7* (p. 876) so that for any mean mixture fraction the scaled variance ranges from 0 to 0.25.

$$\overline{f_s'^2} = 0.25\frac{\overline{f'^2}}{\overline{f}(1-\overline{f})} \qquad \textbf{(17–7)}$$

– For a two-mixture-fraction case, the **Scaled Heat Loss/Gain** is the only available option.

c. (non-adiabatic cases only) Specify whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value** under **Slice by**.

– If you selected **Index**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Table** tab of the **Species Model** dialog box (see *Calculating the Look-Up Tables* (p. 871)). If you specified to hold the enthalpy (**Scaled Heat Loss/Gain**) constant, the enthalpy slice index corresponding to the adiabatic case will be displayed in the **Adiabatic** field.

– If you selected **Value**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

- For a 2D curve on a 3D surface:

a. Specify whether you want to write the plot data to a file by toggling **Write To File** under **Options**.

b. Under **Curve Parameters**, specify the **X-Axis Function** against which the plot variable will be displayed.

– For an adiabatic single-mixture-fraction case, select **Mean Mixture Fraction** ($\overline{f}$), or **Scaled Variance** ($\overline{f'^2}$).

– For a non-adiabatic single-mixture-fraction case, the options will depend on what was selected under **Constant Value of** under **Surface Parameters**, but will include two of the following: **Scaled Heat Loss/Gain** ($\overline{H}$), **Mean Mixture Fraction**, and **Scaled Variance**.

– For a two-mixture-fraction case, select **Fuel Mixture Fraction** (f_{fuel}) or **Secondary Partial Fraction** (p_{sec}).

c. Specify the type of discretization (i.e., how the look-up table data will be sliced) for the variable that is being held constant (under **Constant Value of Mean Mixture Fraction**, **Constant Value of Scaled Variance**, etc.). Note that for non-adiabatic cases, each 3D surface slice contains a full set of 2D slices.

– If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Table** tab of the **Species Model** dialog box (see *Calculating the Look-Up Tables* (p. 871)).

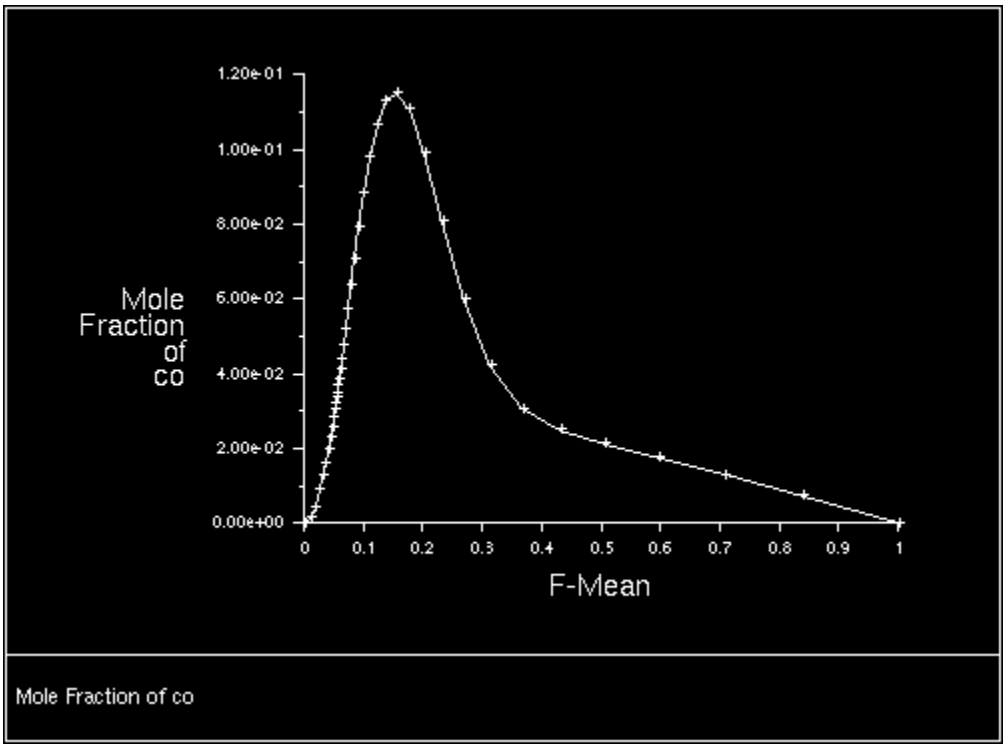
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

4. Write or display the look-up table results. If you have turned on the **Write To File** option for a 2D plot, click **Write** and specify a name for the file in *The **Select File** Dialog Box* (p. 33). Otherwise, click **Plot** or **Display** as appropriate to display a 2D plot or 3D surface in the graphics window.

Figure 17.23 (p. 877) and *Figure 17.24* (p. 878) show examples of 2D plots derived for a very simple hydrocarbon system.

Figure 17.23 Mean Species Fraction Derived From an Equilibrium Chemistry Calculation



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.24 Mean Temperature Derived From an Equilibrium Chemistry Calculation

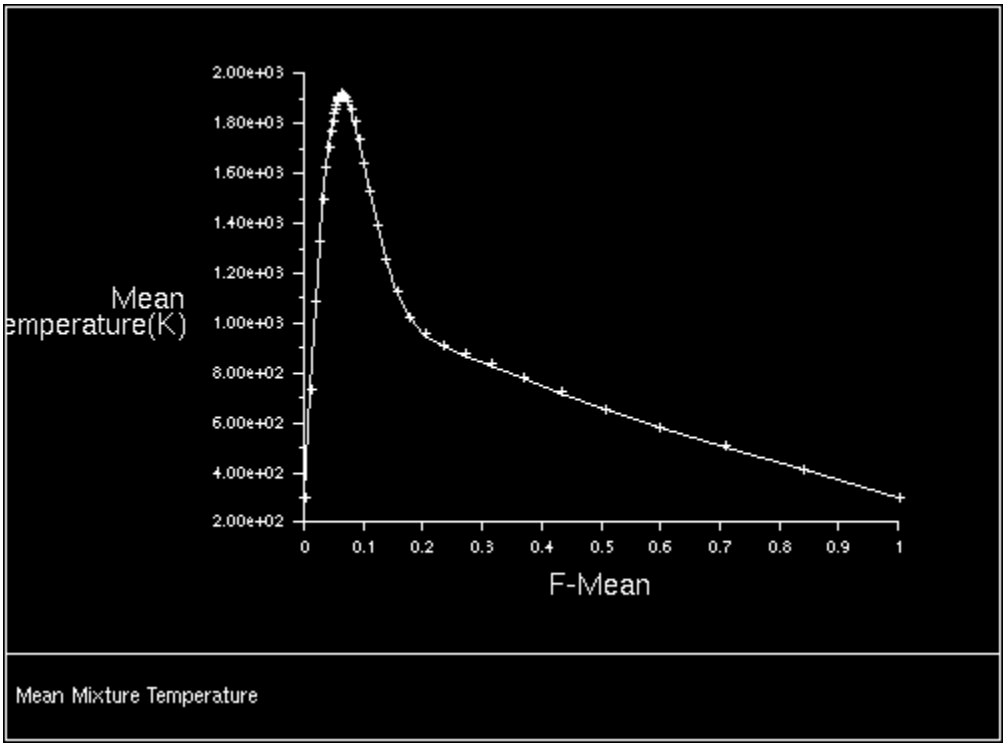
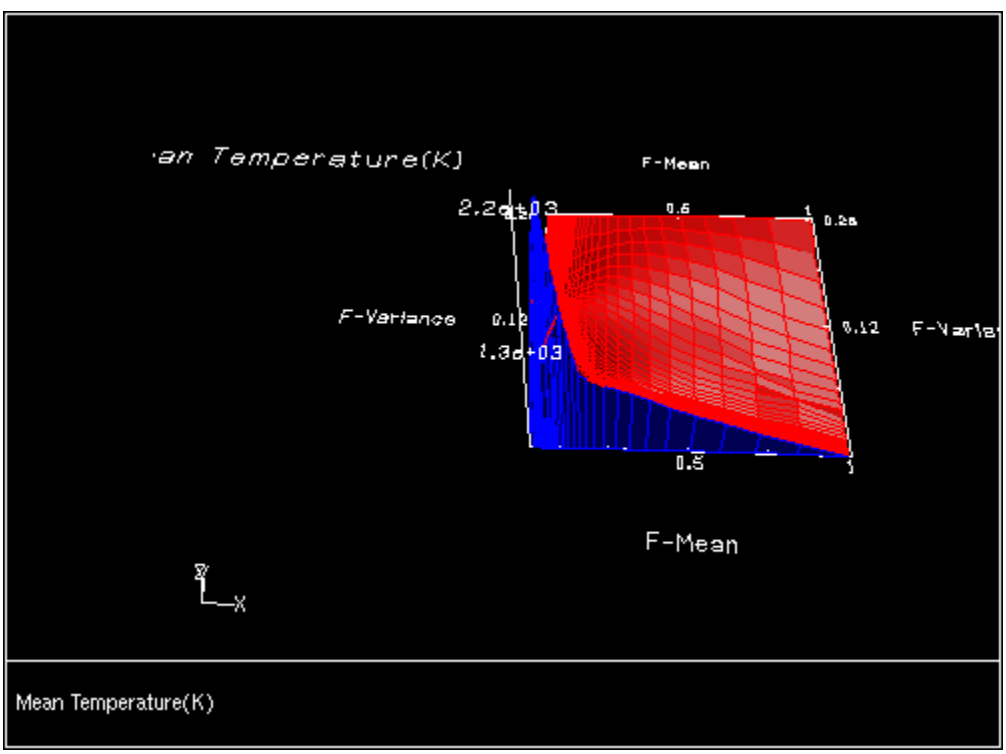


Figure 17.25 (p. 878) shows an example of a 3D surface derived for the same system.

Figure 17.25 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.7.4.1. Files for Flamelet Modeling

In this section, information is provided about the standard flamelet files used for flamelet generation and import.

17.7.4.1.1. Standard Flamelet Files

The data structure for the standard flamelet file format is based on keywords that precede each data section. If any of the keywords in your flamelet data file do not match the supported keywords, you will have to manually edit the file and change the keywords to one of the supported types. (The ANSYS FLUENT flamelet filter is case-insensitive, so you need not worry about capitalization within the keywords.)

The following keywords are supported by the ANSYS FLUENT filter:

- Header section: `HEADER`
- Main body section: `BODY`
- Number of species: `NUMOFSPECIES`
- Number of grid points: `GRIDPOINTS`
- Pressure: `PRESSURE`
- Strain rate: `STRAINRATE`
- Scalar dissipation: `CHI`
- Temperature: `TEMPERATURE` and `TEMP`
- Mass fraction: `MASSFRACTION-`
- Mixture fraction: `Z`

17.7.4.1.1.1. Sample File

A sample flamelet file in the standard format is provided below. Note that not all species are listed in this file.

```
HEADER
STRAINRATE 100.
NUMOFSPECIES 12
GRIDPOINTS 39
PRESSURE 1.
BODY
Z
 0.0000E+00 4.3000E-07 2.1780E-06 1.2651E-05 7.8456E-05
 2.1876E-04 5.9030E-04 9.4701E-04 1.4700E-03 1.8061E-03
 2.1967E-03 2.6424E-03 3.1435E-03 4.3038E-03 5.6637E-03
 8.9401E-03 1.2800E-02 1.7114E-02 2.1698E-02 2.6304E-02
 2.8522E-02 3.0647E-02 3.2680E-02 3.4655E-02 4.2784E-02
 5.2655E-02 6.5420E-02 8.2531E-02 1.0637E-01 1.4122E-01
 1.9518E-01 2.8473E-01 4.4175E-01 6.6643E-01 8.6222E-01
 9.5897E-01 9.9025E-01 9.9819E-01 1.0000E+00
TEMPERATURE
 3.0000E+02 3.0013E+02 3.0085E+02 3.0475E+02 3.2382E+02
 3.5644E+02 4.3055E+02 4.9469E+02 5.8260E+02 6.3634E+02
 6.9655E+02 7.6268E+02 8.3393E+02 9.8775E+02 1.1493E+03
 1.4702E+03 1.7516E+03 1.9767E+03 2.1403E+03 2.2444E+03
 2.2766E+03 2.2962E+03 2.3044E+03 2.3027E+03 2.2164E+03
 2.0671E+03 1.8792E+03 1.6655E+03 1.4355E+03 1.1986E+03
 9.6530E+02 7.5025E+02 5.7496E+02 4.4805E+02 3.6847E+02
 3.2730E+02 3.0939E+02 3.0248E+02 3.0000E+02
MASSFRACTION-H2
 3.2354E-07 7.4290E-07 1.6979E-06 3.8179E-06 8.3038E-06
 1.2219E-05 1.7873E-05 2.1556E-05 2.5872E-05 2.8290E-05
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 3.0888E-05 3.3684E-05 3.6720E-05 4.3768E-05 5.4359E-05
 1.0484E-04 2.6807E-04 6.1906E-04 1.2615E-03 2.3555E-03
 3.1422E-03 4.1281E-03 5.3302E-03 6.7434E-03 1.4244E-02
 2.4296E-02 3.7472E-02 5.5159E-02 7.9788E-02 1.1573E-01
 1.7135E-01 2.6359E-01 4.2527E-01 6.5658E-01 8.5814E-01
 9.5775E-01 9.8996E-01 9.9814E-01 1.0000E+00
MASSFRACTION-CH4
.  .   .    .    .
.  .   .    .    .
.  .   .    .    .
MASSFRACTION-O
 6.8919E-10 2.8720E-09 1.1905E-08 4.8669E-08 2.0370E-07
 5.5281E-07 1.7418E-06 3.6996E-06 8.3107E-06 1.3525E-05
 2.2484E-05 3.8312E-05 6.6385E-05 1.8269E-04 4.4320E-04
 1.4284E-03 2.7564E-03 3.9063E-03 4.3237E-03 3.7141E-03
 3.0916E-03 2.3917E-03 1.7345E-03 1.2016E-03 2.4323E-04
 5.2235E-05 1.1469E-05 2.3011E-06 3.7414E-07 4.2445E-08
 2.7470E-09 8.7551E-11 2.9341E-12 7.0471E-13 0.0000E+00
 7.2143E-14 0.0000E+00 0.0000E+00 0.0000E+00
```

17.7.4.1.1.2. Missing Species

ANSYS FLUENT will check whether all species in the flamelet data file exist in the thermodynamic properties databases `thermo.db`. If any of the species in the flamelet file do not exist, ANSYS FLUENT will issue an error message and halt the flamelet import. If this occurs, you can either add the missing species to the database, or remove the species from the flamelet file.

You should not remove a species from the flamelet data file unless its species concentration is very small (10^{-3} or less) throughout the flamelet profile. If you remove a low-concentration species, you will not have the species concentrations available for viewing in the ANSYS FLUENT calculation, but the accuracy of the ANSYS FLUENT calculation will otherwise be unaffected.

Important

If you choose to remove any species, be sure to also update the number of species (keyword `NUMOFSPECIES`) in the flamelet data file, to reflect the loss of any species you have removed from the file.

If a species with relatively large concentration is missing from the ANSYS FLUENT thermodynamic databases, you will have to add it. Removing a high-concentration species from the flamelet file is not recommended.

17.7.5. Setting Up the Inert Model

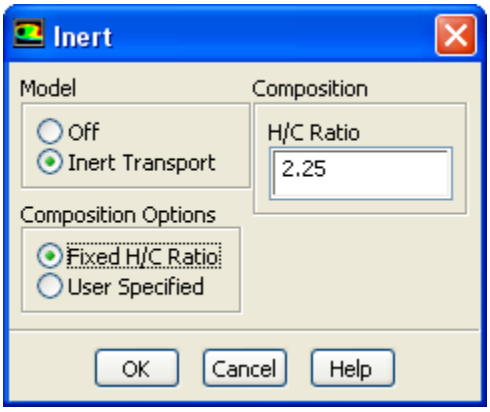
This section describes how to set up and apply the inert model. For a discussion about the theory, refer to Using the Non-Premixed Model with the Inert Model in the Theory Guide.

To enable the inert model, make sure that the non-premixed or partially premixed model is selected in the **Species Model** dialog box, or that a PDF file is read. Refer to *Steps in Using the Non-Premixed Model* (p. 839) and *Modeling Partially Premixed Combustion* (p. 905) to learn more about these models.

Models → **Inert** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.26 The Inert Model Dialog Box



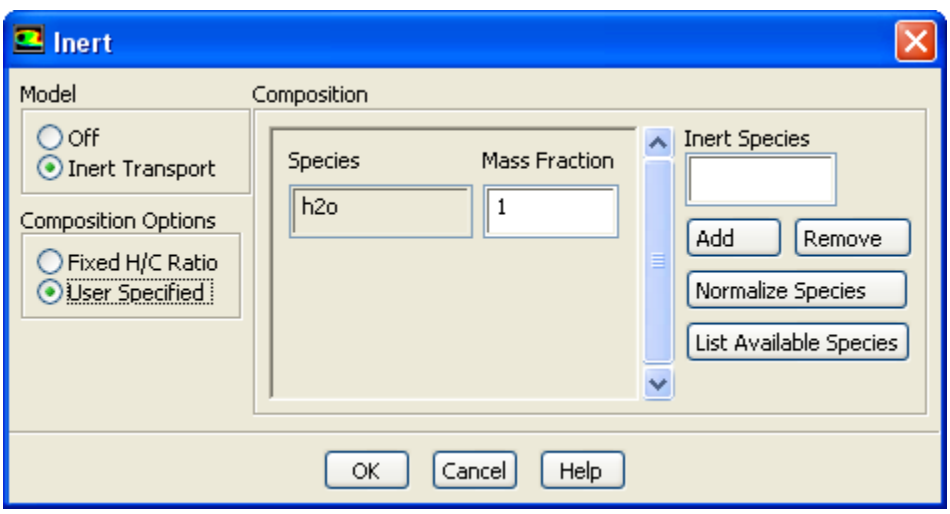
The **Inert** dialog box will be displayed (*Figure 17.26* (p. 881)). To enable this model, select **Inert Transport**. The following steps will walk you through setting up the inert model:

1. Select **Fixed H/C Ratio** as the **Composition Option** if the hydrogen to carbon ratio is known. For example, for methane (CH_4) enter 4 for **H/C Ratio**.

   Setting the H/C ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel with air, and the only products of the combustion are CO_2, H_2O and N_2.

2. Select **User Specified** as the **Composition Option** if you want to specify an arbitrary composition for the inert stream, as shown in *Figure 17.27* (p. 881).

   Figure 17.27 The Inert Model Dialog Box



   You can specify your composition stream by adding or removing species if your stream is composed of species other than the default species list.

   a. To add species to the **Species** list, type the chemical formula under **Inert Species** and click **Add**.

   b. Enter the **Mass Fraction** of the newly added species. Continue in this manner until all of the inert species you want to include are shown in the **Species** list.

   c. Make sure the sum of the mass fractions add up to 1. ANSYS FLUENT will normalize the species mass fractions for you when you click **Normalize Species**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. To remove a species from the list, type the chemical formula under **Inert Species** and click **Remove**.

e. To print a list of all species in the thermodynamic database file (`thermo.db`) in the console window, click **List Available Species**.

17.7.5.1. Setting Boundary Conditions for Inert Transport

You will need to set appropriate boundary conditions at flow inlets and exits for the inert tracer mass fraction, Y_I. The tracer species mass fraction must be between zero and one, with the value of one meaning that all of the material entering the domain comes from the inert stream. The values for flow boundaries are set in the **Inert Stream** field of the inlet boundary condition dialog boxes, under the **Species** tab.

17.7.5.2. Initializing the Inert Stream

The main assumption of the inert model is that the composition of the inert stream does not change with combustion. For some dilutants, this is a very reasonable assumption, however, it is not valid for rich combustion where there is fuel in the exhaust stream. For cases where there is fuel or oxidizer left in the exhaust gas, accurate results will depend upon the user taking the fuel or oxidizer species into account when setting initial conditions.

17.7.5.2.1. Inert Fraction

Initialization of the inert mass fraction Y_I is done in the same way as other variables: by entering in the appropriate value in the **Solution Initialization** task page. Another option for initialization is to patch the value of Y_I in a region of the domain. When the value of Y_I is patched in this way, ANSYS FLUENT automatically recalculates the enthalpy field for the current temperature field in order to account for the change in composition.

See *Patching Values in Selected Cells* (p. 1242) for details about patching values of solution variables.

17.7.5.2.2. Inert Composition

For combustion calculations which burn a hydrocarbon fuel, ANSYS FLUENT provides a straightforward way of setting the initial composition of the inert stream. The inert composition can be set by assuming a ratio of hydrogen to carbon in the following overall oxidation reaction (from Heywood [*32*] (p. 2210)):

$$C_aH_b+\left(a+\frac{b}{4}\right)(O_2+3.773N_2)=aCO_2+\frac{b}{2}H_2O+3.773\left(a+\frac{b}{4}\right)N_2 \qquad \textbf{(17–8)}$$

which can be rewritten in terms of the ratio of hydrogen to carbon atoms ($y=b/a$) in the fuel as

$$CH_y+\left(1+\frac{y}{4}\right)(O_2+3.773N_2)=CO_2+\frac{y}{2}H_2O+3.773\left(1+\frac{y}{4}\right)N_2 \qquad \textbf{(17–9)}$$

If *Equation 17–9* (p. 882) is solved for the mass fractions of CO_2, H_2O and N_2, the following relations are obtained:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$
\begin{aligned}
Y_{CO_2} &= \frac{44.01}{y_{tot}} \\
Y_{H_2O} &= \frac{9.01y}{y_{tot}} \\
Y_{N_2} &= \frac{26.56192y+106.24768}{y_{tot}}
\end{aligned}
\tag{17–10}
$$

where

$$y_{tot}=35.57192y+150.25768$$

Setting the H/C ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel, and the only products of the combustion are CO_2, H_2O and N_2. An arbitrary composition for the inert stream can also be specified in the interface.

17.8. Defining Non-Premixed Boundary Conditions

For information about defining non-premixed boundary conditions, please see the following sections:

17.8.1. Input of Mixture Fraction Boundary Conditions
17.8.2. Diffusion at Inlets
17.8.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities

17.8.1. Input of Mixture Fraction Boundary Conditions

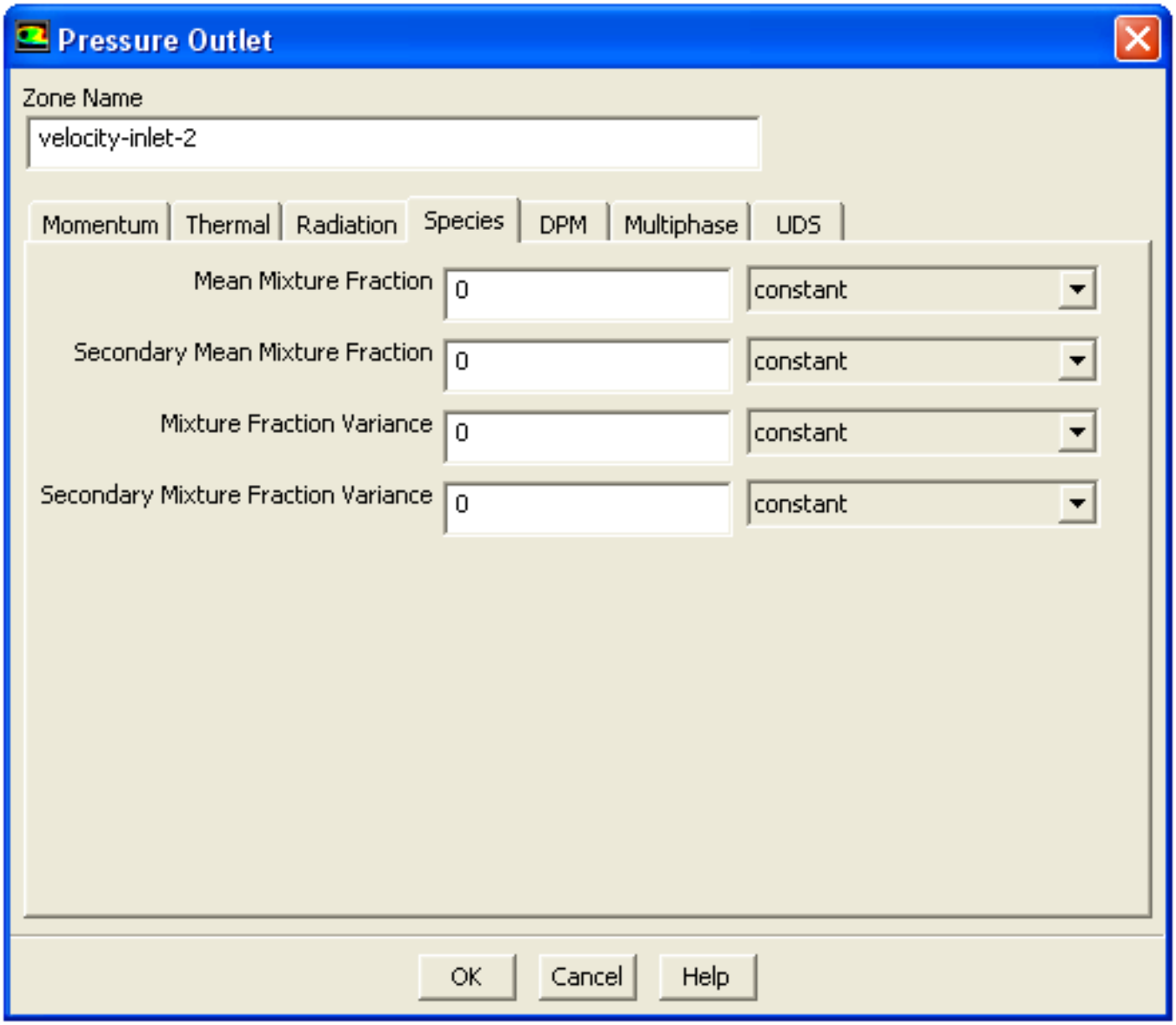
When the non-premixed combustion model is used, flow boundary conditions at inlets and exits (i.e., velocity or pressure, turbulence intensity) are defined in the usual way. Species mass fractions at inlets are not required.

Instead, you define values for the mean mixture fraction, $\overline{f}$, and the mixture fraction variance, $\overline{f'^2}$, at inlet boundaries. (For problems that include a secondary stream, you will define boundary conditions for the mean secondary partial fraction and its variance as well as the mean fuel mixture fraction and its variance.) These inputs provide boundary conditions for the conservation equations you will solve for these quantities. The inlet values are supplied in the boundary conditions task page, under the available tabs, for the selected inlet boundary (e.g., *Figure 17.28* (p. 884)).

Boundary Conditions

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.28 The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions



Click the **Species** tab and input the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions). In general, the inlet value of the mean fractions will be 1.0 or 0.0 at flow inlets: the mean fuel mixture fraction will be 1.0 at fuel stream inlets and 0.0 at oxidizer or secondary stream inlets; the mean secondary mixture fraction will be 1.0 at secondary stream inlets and 0.0 at fuel or oxidizer inlets. The fuel or secondary mixture fraction will lie between 0.0 and 1.0 only if you are modeling flue gas recycle, as illustrated in Figure 8.15: "Using the Non-Premixed Model with Flue Gas Recycle" and discussed in Definition of the Mixture Fraction in the Theory Guide. The fuel or secondary mixture fraction variance can usually be taken as zero at inlet boundaries.

17.8.2. Diffusion at Inlets

In some cases, you may wish to include the diffusive transport of mixture fraction through the inlets of your domain. You can do this by enabling inlet mixture-fraction diffusion. By default, ANSYS FLUENT excludes the diffusion flux of mixture fraction at inlets. To enable inlet diffusion, use either the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
define/models/ species/inlet-diffusion?
```

text command, or the *Species Model Dialog Box* (p. 1681)

Models → **Species** → **Edit...**

and enable the **Inlet Diffusion** option.

17.8.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities

If your model is non-adiabatic, you should input the **Temperature** at the flow inlets. Recall that the inlet temperatures were requested during the table construction in the **Chemistry** tab of the **Species Model** dialog box, and were used in the construction of the look-up tables. The inlet temperatures for each fuel, oxidizer, and secondary inlet in your non-adiabatic model should be defined, in addition, as boundary conditions in ANSYS FLUENT. It is acceptable for the inlet temperature boundary conditions to differ slightly from those you input for the look-up table calculations. If the inlet temperatures differ significantly from those in the **Chemistry** tab, however, your look-up tables may provide inaccurate interpolation. This is because the discrete points in the look-up tables were clustered around a finite range of the temperatures defined in the **Chemistry** tab, and your input in the ANSYS FLUENT inlet boundary condition may fall outside this range.

Wall thermal boundary conditions should also be defined for non-adiabatic non-premixed combustion calculations. You can use any of the standard conditions available in ANSYS FLUENT, including specified wall temperature, heat flux, external heat transfer coefficient, or external radiation. If radiation is to be included within the domain, the wall emissivity should be defined as well. See *Thermal Boundary Conditions at Walls* (p. 309) for details about thermal boundary conditions at walls.

17.9. Defining Non-Premixed Physical Properties

When you use the non-premixed combustion model, the material used for all fluid zones is automatically set to **pdf-mixture**. This material is a special case of the mixture material concept discussed in *Mixture Materials* (p. 800). The constituent species of this mixture are the species that were calculated in the PDF look-up table creation; you cannot change them directly. When the non-premixed model is used, heat capacities, molecular weights, and enthalpies of formation for each species considered are extracted from the chemical database, so you will not modify any properties for the constituent species in the PDF mixture. For the PDF mixture itself, the mean density and mean specific heat are determined from the look-up tables.

The physical property inputs for a non-premixed combustion problem are therefore only the transport properties (viscosity, thermal conductivity, etc.) for the PDF mixture. To set these in the *Create/Edit Materials Dialog Box* (p. 1736), choose **mixture** as the **Material Type**, **pdf-mixture** (the default, and only choice) in the **Mixture Materials** list, and set the desired values for the transport properties.

Materials

See *Physical Properties* (p. 385) for details about setting physical properties. The transport properties in a non-premixed combustion problem can be defined as functions of temperature, if desired, but not as functions of composition. In practice, since turbulence effects will dominate, it will be of little benefit to include even the temperature dependence of the laminar transport properties.

If you are modeling radiation heat transfer, you will also input radiation properties, as described in *Radiation Properties* (p. 435). Composition-dependent absorption coefficients (using the WSGGM) are allowed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.10. Solution Strategies for Non-Premixed Modeling

The non-premixed model setup and solution procedure in ANSYS FLUENT differs slightly for single- and two-mixture-fraction problems. Below, an overview of each approach is provided. Note that your ANSYS FLUENT case file must always meet the restrictions listed for the non-premixed modeling approach in Restrictions on the Mixture Fraction Approach in the Theory Guide. In this section, details are provided regarding the problem definition and calculation procedures you follow in ANSYS FLUENT.

For additional information, please see the following sections:

17.10.1. Single-Mixture-Fraction Approach
17.10.2. Two-Mixture-Fraction Approach
17.10.3. Starting a Non-Premixed Calculation From a Previous Case File
17.10.4. Solving the Flow Problem

17.10.1. Single-Mixture-Fraction Approach

For a single-mixture-fraction system, when you have completed the calculation of the PDF look-up tables, you are ready to begin your reacting flow simulation. In ANSYS FLUENT, you will solve the flow field and predict the spatial distribution of $\overline{f}$ and $\overline{f'^2}$ (and $\overline{H}$ if the system is non-adiabatic or $\overline{\chi}$ if the system is based on laminar flamelets). ANSYS FLUENT will obtain the corresponding values of temperature and individual chemical species mass fractions from the look-up tables.

17.10.2. Two-Mixture-Fraction Approach

When a secondary stream is included, ANSYS FLUENT will solve transport equations for the mean secondary partial fraction (p_{sec}) and its variance in addition to the mean fuel mixture fraction and its variance. ANSYS FLUENT will then look up the instantaneous values for temperature, density, and individual chemical species in the look-up tables, compute the PDFs for the fuel and secondary streams, and calculate the mean values for temperature, density, and species.

Note that in order to avoid both inaccuracies and unnecessarily slow calculation times, it is important for you to view your temperature and species tables in ANSYS FLUENT to ensure that they are adequately but not excessively resolved.

17.10.3. Starting a Non-Premixed Calculation From a Previous Case File

You can read a previously defined ANSYS FLUENT case file as a starting point for your non-premixed combustion modeling. If this case file contains inputs that are incompatible with the current non-premixed combustion model, ANSYS FLUENT will alert you when the non-premixed model is turned on and it will turn off those incompatible models. For example, if the case file includes species that differ from those included in the PDF file created by ANSYS FLUENT, these species will be disabled. If the case file contains property descriptions that conflict with the property data in the chemical database, these property inputs will be ignored.

Important

PDF files created by prePDF 2 or older are not supported by this version of ANSYS FLUENT. The files generated by PrePDF version 3 or newer, are fully compatible.

In the **Species Model** dialog box, select **Non-Premixed Combustion** under the **Model** heading. When you click **OK** in the **Species Model** dialog box, *The **Select File** Dialog Box* (p. 33) will immediately appear,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

prompting you for the name of the PDF file containing the look-up tables created in a previous ANSYS FLUENT session. (The PDF file is the file you saved using the **File/Write/PDF...** menu item after computing the look-up tables.) ANSYS FLUENT will indicate that it has successfully read the specified PDF file:

```
Reading "/home/mydirectory/adiabatic.pdf"...
read 5 species (binary c, adiabatic fluent)
pdf file successfully read.
Done.
```

After you read in the PDF file, ANSYS FLUENT will inform you that some material properties have changed. You can accept this information; you will be updating properties later on.

You can read in an altered PDF file at any time by using the **File/Read/PDF...** menu item.

Important

Recall that the non-premixed combustion model is available only when you used the pressure-based solver; it cannot be used with the density-based solvers. Also, the non-premixed combustion model is available only when turbulence modeling is active.

If you are modeling a non-adiabatic system and you wish to include the effects of compressibility, re-open the **Species Model** dialog box and turn on **Compressibility Effects** under **PDF Options**. This option tells ANSYS FLUENT to update the density according to *Equation 17–1* (p. 845). When the non-premixed combustion model is active, you can enable compressibility effects only in the **Species Model** dialog box. For other models, you will specify compressible flow (**ideal-gas**, **boussinesq**, etc.) in the **Create/Edit Materials** dialog box. See *Specifying the Operating Pressure for the System* (p. 845) and *Tuning the PDF Parameters for Two-Mixture-Fraction Calculations* (p. 888) for more information about compressibility effects.

17.10.3.1. Retrieving the PDF File During Case File Reads

The PDF filename is specified to ANSYS FLUENT only once. Thereafter, the filename is stored in your ANSYS FLUENT case file and the PDF file will be automatically read into ANSYS FLUENT whenever the case file is read. ANSYS FLUENT will remind you that it is reading the PDF file after it finishes reading the rest of the case file by reporting its progress in the text (console) window.

Note that the PDF filename stored in your case file may not contain the full name of the directory in which the PDF file exists. The full directory name will be stored in the case file only if you initially read the PDF file through the GUI (or if you typed in the directory name along with the filename when using the text interface). In the event that the full directory name is absent, the automatic reading of the PDF file may fail (since ANSYS FLUENT does not know which directory to look in for the file), and you will need to manually specify the PDF file. The safest approaches are to use the GUI when you first read the PDF file or to supply the full directory name when using the text interface.

17.10.4. Solving the Flow Problem

The next step in the non-premixed combustion modeling process in ANSYS FLUENT is the solution of the mixture fraction and flow equations. First, initialize the flow. By default, the mixture fraction and its variance have initial values of zero, which is the recommended value; you should generally not set non-zero initial values for these variables. See *Initializing the Solution* (p. 1239) for details about solution initialization.

Solution Initialization

Next, begin calculations in the usual manner.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Calculation

During the calculation process, ANSYS FLUENT reports residuals for the mixture fraction and its variance in the `fmean` and `fvar` columns of the residual report:

```
iter       cont       x-vel       y-vel          k     epsilon       fmean       fvar
  28    1.57e-3     4.92e-4     4.80e-4    2.68e-2     2.59e-3     9.09e-1    1.17e+0
  29    1.42e-3     4.43e-4     4.23e-4    2.48e-2     2.30e-3     8.89e-1    1.15e+0
  30    1.28e-3     3.98e-4     3.75e-4    2.29e-2     2.04e-3     8.88e-1    1.14e+0
```

(For two-mixture-fraction calculations, columns for `psec` and `pvar` will also appear.)

17.10.4.1. Under-Relaxation Factors for PDF Equations

The transport equations for the mean mixture fraction and mixture fraction variance are quite stable and high, under-relaxation can be used when solving them. By default, an under-relaxation factor of 1 is used for the mean mixture fraction (and secondary partial fraction) and 0.9 for the mixture fraction variance (and secondary partial fraction variance). If the residuals for these equations are increasing, you should consider decreasing these under-relaxation factors, as discussed in *Setting Under-Relaxation Factors* (p. 1217).

17.10.4.2. Density Under-Relaxation

One of the main reasons a combustion calculation can have difficulty converging is that large changes in temperature cause large changes in density, which can, in turn, cause instabilities in the flow solution. ANSYS FLUENT allows you to under-relax the change in density to alleviate this difficulty. The default value for density under-relaxation is 1, but if you encounter convergence trouble you may wish to reduce this to a value between 0.5 and 1 (in the **Solution Controls** task page).

17.10.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations

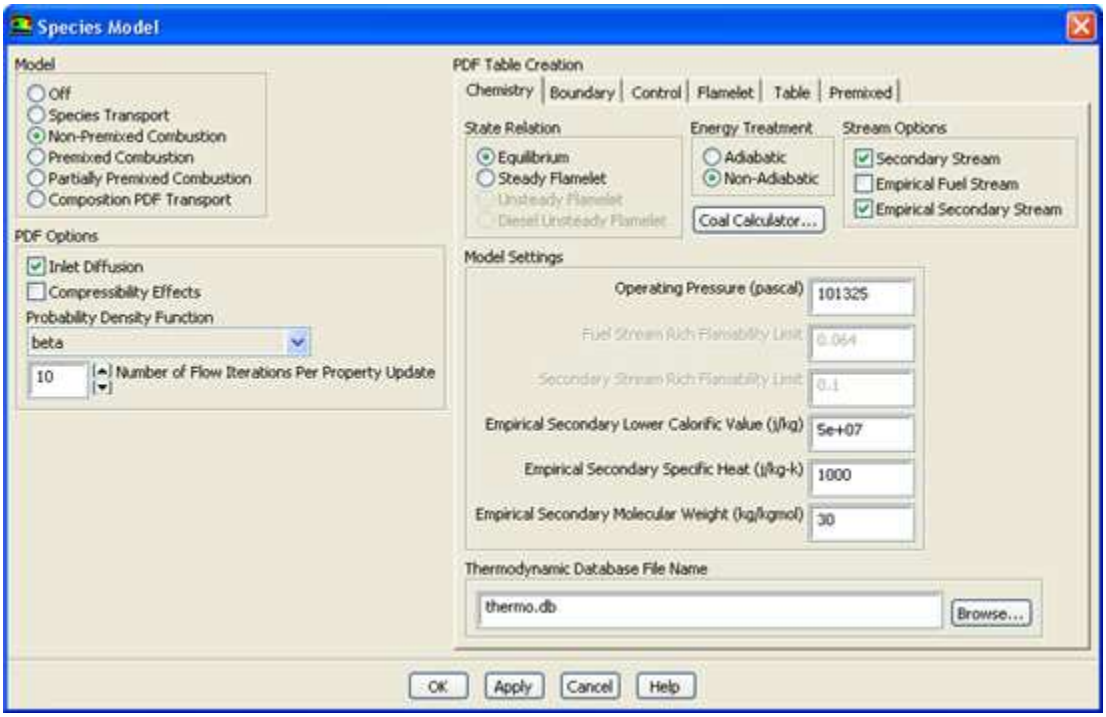
For cases that include a secondary stream, the PDF integrations are performed inside ANSYS FLUENT.

The parameters for these integrations are defined in the *Species Model Dialog Box* (p. 1681) (*Figure 17.29* (p. 889)).

Models → **Species** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.29 The Species Model Dialog Box for a Two-Mixture-Fraction Calculation



The parameters are as follows:

Compressibility Effects

(non-adiabatic systems only) tells ANSYS FLUENT to update the density, temperature, species mass fraction, and enthalpy from the PDF tables to account for the varying pressure of the system.

Probability Density Function

specifies which type of PDF should be used. You can pick either **double delta** or **beta** in the drop-down list. The **double delta** PDF has the advantage of being faster than the **beta** PDF, but is theoretically less accurate.

Number of Flow Iterations Per Property Update

specifies how often the flow properties (density, temperature, etc.) are updated from the look-up table. Remember that when you are calculating two mixture fractions, the updating of properties includes computation of the PDFs and can be CPU-intensive.

For simulations involving non-adiabatic multiple strained flamelets, looking up the four-dimensional PDF tables can be CPU-intensive. In such cases, the **Number of Flow Iterations Per Property Update** controls the updating of flow properties such as density.

For the Eulerian unsteady laminar flamelet model, a marker probability equation is solved in an unsteady mode. Residuals for `ufla-prob` will be displayed.

17.11. Postprocessing the Non-Premixed Model Results

The final step in the non-premixed combustion modeling process is the postprocessing of species concentrations and temperature data from the mixture fraction and flow-field solution data. The following variables are of particular interest:

- **Mean Mixture Fraction** (in the **Pdf...** category)
- **Secondary Mean Mixture Fraction** (in the **Pdf...** category)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mixture Fraction Variance** (in the **Pdf...** category)
- **Secondary Mixture Fraction Variance** (in the **Pdf...** category)
- **Fvar Prod** (in the **Pdf...** category, which is the production term in the mixture fraction variance transport equation)
- **Fvar2 Prod** (in the **Pdf...** category)
- **Scalar Dissipation** (in the **Pdf...** category)
- **PDF Table Adiabatic Enthalpy** (in the **Pdf...** category)
- **PDF Table Heat Loss/Gain** (in the **Pdf...** category)
- **Mass fraction of (species-n)** (in the **Species...** category)
- **Mole fraction of (species-n)** (in the **Species...** category)
- **Molar Concentration of (species-n)** (in the **Species...** category)
- **RMS (species-n) Mass Fraction** (in the **Species...** category)
- **Static Temperature** (in the **Temperature...** category)
- **RMS Temperature** (in the **Temperature...** category)
- **Enthalpy** (in the **Temperature...** category)
- **Mean Probability** (in the **Unsteady Flamelet...** category)
- **Mean Temperature** (in the **Unsteady Flamelet...** category)
- **Mean Mass Fraction of (species-n)** (in the **Unsteady Flamelet...** category)

Important

For the unsteady laminar flamelet model, mean species mass fractions are displayed for the first fifty species in the flamelet kinetic mechanism.

These quantities can be selected for display in the indicated category of the variable-selection drop-down list that appears in postprocessing dialog boxes. See *Field Function Definitions* (p. 1529) for their definitions.

In all cases, the species concentrations are derived from the mixture fraction/variance field using the look-up tables. Note that temperature and enthalpy can be postprocessed even when your ANSYS FLUENT model is an adiabatic non-premixed combustion simulation in which you have not solved the energy equation. In both the adiabatic and non-adiabatic cases, the temperature is derived from the look-up table.

Figure 17.30 (p. 891) and *Figure 17.31* (p. 891) illustrate typical results for a methane diffusion flame modeled using the non-premixed approach.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.30 Predicted Contours of Mixture Fraction in a Methane Diffusion Flame

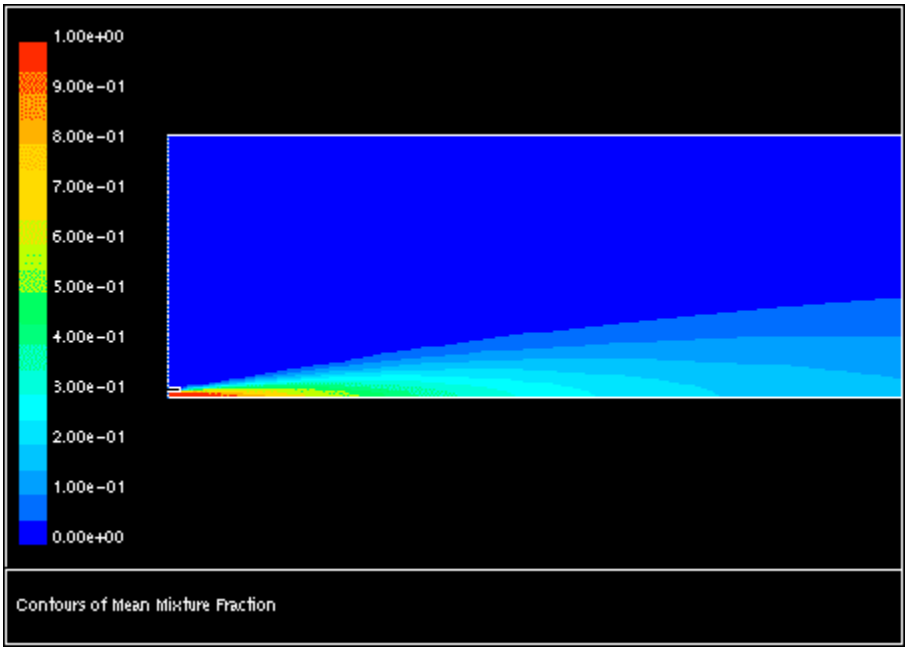
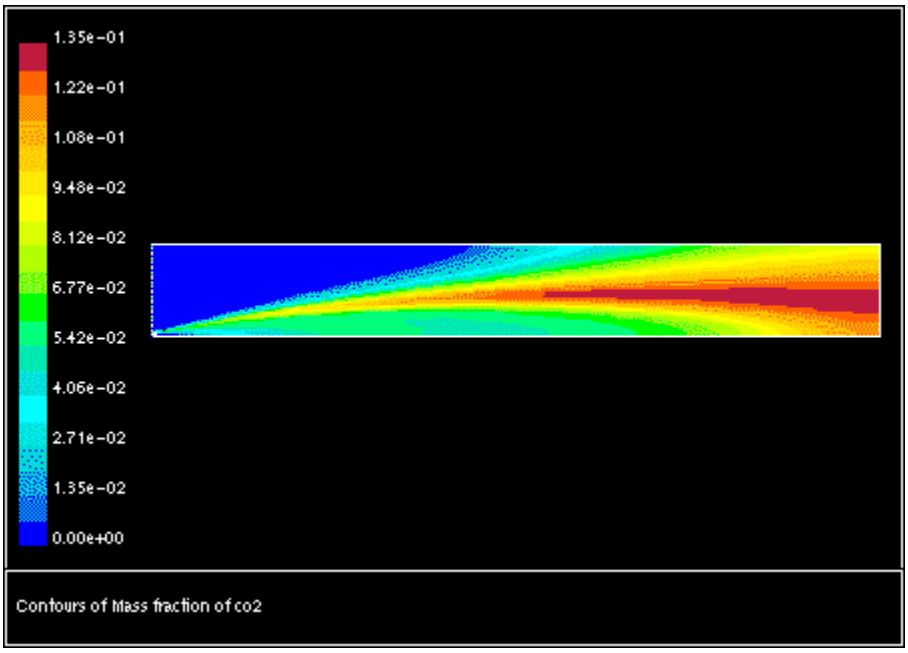


Figure 17.31 Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model



For additional information, please see the following sections:

17.11.1. Postprocessing for Inert Calculations

17.11.1. Postprocessing for Inert Calculations

ANSYS FLUENT provides several additional reporting options for post processing calculations with the inert model. You can generate graphical plots or alphanumeric reports of the same items that are available with the non premixed or partially premixed models, and in addition the following variables are available:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Inert Mass Fraction**
- **Inert Specific Heat**
- **Inert Density**
- **Inert Enthalpy**
- **Pdf Enthalpy**
- **Pdf Fmean**
- **Pdf Fvar**
- **Mass Fraction of Inert** Y_i

where the mass fraction of the ith inert species, Y_i, is calculated as

$$Y_i = Y_I Y_i^I$$

where Y_I is the inert tracer and Y_i^I is the mass fraction of the ith inert species defined in the **Inert** dialog box.

Note these variables appear in the **Inert...** category.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 18: Modeling Premixed Combustion

ANSYS FLUENT has several models to simulate premixed turbulent combustion. For theoretical background on this model, see "Premixed Combustion" in the Theory Guide. Information about using this model is provided in the following sections:

18.1. Overview and Limitations
18.2. Using the Premixed Combustion Model
18.3. Setting Up the C-Equation and G-Equation Models
18.4. Setting Up the Extended Coherent Flame Model
18.5. Postprocessing for Premixed Combustion Calculations

18.1. Overview and Limitations

For additional information, please see the following sections:

18.1.1. Overview
18.1.2. Limitations

18.1.1. Overview

In premixed combustion, fuel and oxidizer are mixed at the molecular level prior to ignition. Combustion occurs as a flame front propagating into the unburnt reactants. Examples of premixed combustion include aspirated internal combustion engines, lean-premixed gas turbine combustors, and gas-leak explosions.

Premixed combustion is much more difficult to model than non-premixed combustion. The reason for this is that premixed combustion usually occurs as a thin, propagating flame that is stretched and contorted by turbulence. For subsonic flows, the overall rate of propagation of the flame is determined by both the laminar flame speed and the turbulent eddies. The laminar flame speed is determined by the rate that species and heat diffuse upstream into the reactants and burn. To capture the laminar flame speed, the internal flame structure would need to be resolved, as well as the detailed chemical kinetics and molecular diffusion processes. Since practical laminar flame thicknesses are of the order of millimeters or smaller, resolution requirements are usually unaffordable.

The effect of turbulence is to wrinkle and stretch the propagating laminar flame sheet, increasing the sheet area and, in turn, the effective flame speed. The large turbulent eddies tend to wrinkle and corrugate the flame sheet, while the small turbulent eddies, if they are smaller than the laminar flame thickness, may penetrate the flame sheet and modify the laminar flame structure.

Non-premixed combustion, in comparison, can be greatly simplified to a mixing problem (see the mixture fraction approach in Introduction in the Theory Guide). The essence of premixed combustion modeling lies in capturing the turbulent flame speed, which is influenced by both the laminar flame speed and the turbulence.

In premixed flames, the fuel and oxidizer are intimately mixed before they enter the combustion device. Reaction then takes place in a combustion zone that separates unburnt reactants and burnt combustion products. Partially premixed flames exhibit the properties of both premixed and diffusion flames. They occur when an additional oxidizer or fuel stream enters a premixed system, or when a diffusion flame becomes lifted off the burner so that some premixing takes place prior to combustion.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Premixed and partially premixed flames can be modeled using ANSYS FLUENT's finite-rate/eddy-dissipation formulation (see *Modeling Species Transport and Finite-Rate Chemistry* (p. 799)). If finite-rate chemical kinetic effects are important, the Laminar Finite-Rate model (see The Laminar Finite-Rate Model in the Theory Guide), the EDC model (see The Eddy-Dissipation-Concept (EDC) Model in the Theory Guide) or the composition PDF transport model (see *Modeling a Composition PDF Transport Problem* (p. 911)) can be used. For information about ANSYS FLUENT's partially premixed combustion model, see *Modeling Partially Premixed Combustion* (p. 905). If the flame is perfectly premixed, so only one stream at one equivalence ratio enters the combustor, it is possible to use the premixed combustion model, as described in this chapter.

18.1.2. Limitations

The following limitations apply to the premixed combustion model:

- You must use the pressure-based solver. The premixed combustion model is not available with the density-based solver.
- The premixed combustion model is valid only for turbulent, subsonic flows. These types of flames are called deflagrations. Explosions, also called detonations, where the combustible mixture is ignited by the heat behind a shock wave, can be modeled with the finite-rate model using the density-based solver. See *Modeling Species Transport and Finite-Rate Chemistry* (p. 799) for information about the finite-rate model.
- The premixed combustion model cannot be used in conjunction with the pollutant (i.e., soot and NOx) models. However, a perfectly premixed system can be modeled with the partially premixed model (see *Modeling Partially Premixed Combustion* (p. 905)), which can be used with the pollutant models.
- You cannot use the premixed combustion model to simulate reacting discrete-phase particles, since these would result in a partially premixed system. Only inert particles can be used with the premixed combustion model.
- The G-Equation model can only be used with the unsteady solver because it tracks the flame front in time. However, RANS solutions which tend to a steady-state can be modeled by evolving them in time until the solution is stationary.

18.2. Using the Premixed Combustion Model

The procedure for setting up and solving a premixed combustion model is outlined below, and then described in detail. Remember that only the steps that are pertinent to premixed combustion modeling are shown here. For information about inputs related to other models that you are using in conjunction with the premixed combustion model, see the appropriate sections for those models.

1. Enable the premixed turbulent combustion model and set the related parameters.

   Models → **Species** → **Edit...**

2. Define the physical properties for the unburnt and burnt material in the domain.

   Materials → **Create/Edit...**

3. Set the value of the progress variable c at flow inlets and exits.

   Boundary Conditions

4. Initialize the value of the progress variable.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Initialization → **Patch...**

5. Solve the problem and perform postprocessing.

Important

If you are interested in computing the concentrations of individual species in the domain, you can use the partially premixed model described in *Modeling Partially Premixed Combustion* (p. 905). Alternatively, compositions of the unburnt and burnt mixtures can be obtained from external analyses using equilibrium or kinetic calculations.

For additional information, please see the following sections:
18.2.1. Enabling the Premixed Combustion Model
18.2.2. Choosing an Adiabatic or Non-Adiabatic Model

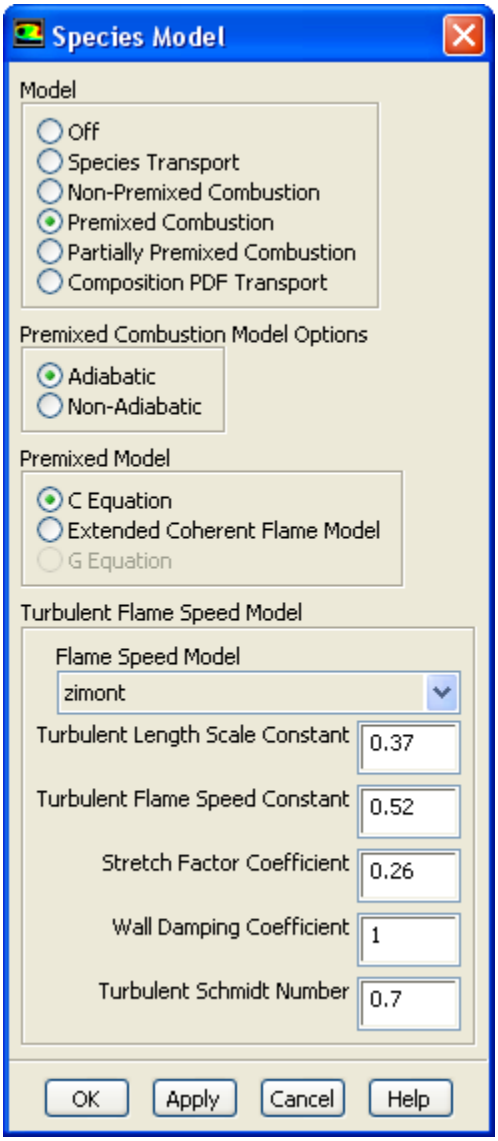
18.2.1. Enabling the Premixed Combustion Model

To enable the premixed combustion model, select **Premixed Combustion** under **Model** in the *Species Model Dialog Box* (p. 1681) (*Figure 18.1* (p. 896)).

Models → **Species** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.1 The Species Model Dialog Box for Premixed Combustion



When you enable **Premixed Combustion**, the dialog box will expand to show the relevant inputs.

18.2.2. Choosing an Adiabatic or Non-Adiabatic Model

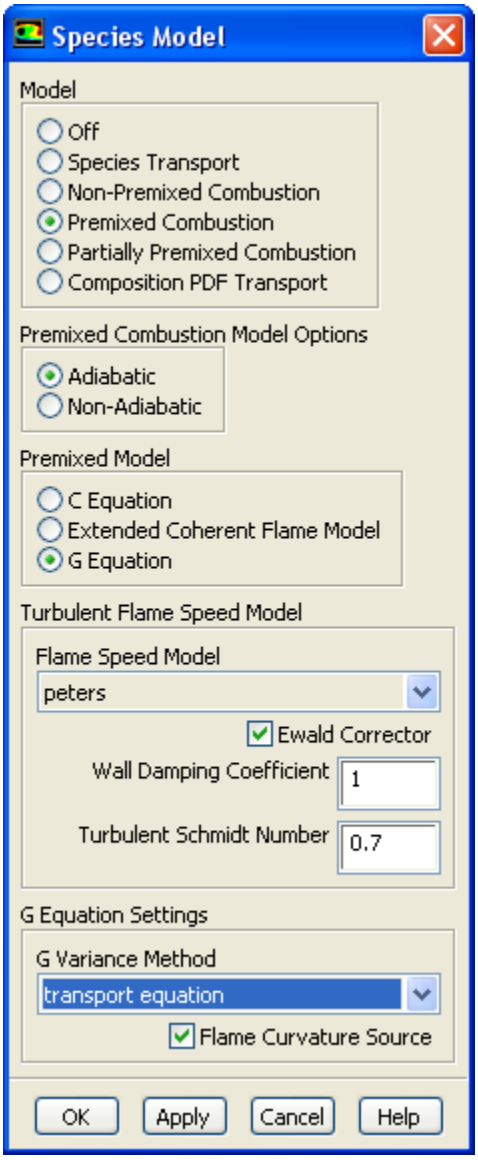
Under **Premixed Combustion Model Options** in the *Species Model Dialog Box* (p. 1681), choose either **Adiabatic** (the default) or **Non-Adiabatic**. This choice will affect only the calculation method used to determine the temperature (either Equation 9–64 or Equation 9–65 in the Theory Guide).

18.3. Setting Up the C-Equation and G-Equation Models

When **C Equation** or **G Equation** is selected as the **Premixed Model**, you can then choose to use the **zimont** or **peters Flame Speed Model** (described in Turbulent Flame Speed Models). You can also modify a number of model constants, as described in.*Modifying the Constants for the Zimont Flame Speed Model* (p. 898) and *Modifying the Constants for the Peters Flame Speed Model* (p. 898).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.2 The Species Model Dialog Box for the G-Equation Model



For a non-adiabatic premixed combustion model, note that the value you specify for the **Turbulent Schmidt Number** will also be used as the Prandtl number for energy. (The **Energy Prandtl Number** will therefore not appear in the **Viscous Model** dialog box for non-adiabatic premixed combustion models.) These parameters control the level of diffusion for the progress variable and for energy. Since the progress variable is closely related to energy (because the flame progress results in heat release), it is important that the transport equations use the same level of diffusion.

For additional information, please see the following sections:

18.3.1. Modifying the Constants for the Zimont Flame Speed Model
18.3.2. Modifying the Constants for the Peters Flame Speed Model
18.3.3. Additional Options for the G-Equation Model
18.3.4. Defining Physical Properties for the Unburnt Mixture
18.3.5. Setting Boundary Conditions for the Progress Variable
18.3.6. Initializing the Progress Variable

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18.3.1. Modifying the Constants for the Zimont Flame Speed Model

In general, you will not need to modify the constants used in the equations presented in C-Equation Model Theory in the Theory Guide. The default values are suitable for a wide range of premixed flames.

You can set the **Turbulent Length Scale Constant** (C_D in Equation 9–11), **Turbulent Flame Speed Constant** (A in Equation 9–9), the **Stretch Factor Coefficient** (μ_{str} in Equation 9–16), the **Turbulent Schmidt Number** (Sc_t in Equation 9–1), and the **Wall Damping Coefficient** (α_w in Equation 9–19).

18.3.2. Modifying the Constants for the Peters Flame Speed Model

The Peters turbulent flame speed model constants, as described in Peters Flame Speed Model in the Theory Guide, are not available in the GUI since they are generally suitable for a wide range of premixed flames. They can, however, be accessed from the TUI.

The **Ewald Corrector** is enabled by default and described in Peters Flame Speed Model. The **Turbulent Schmidt Number** (Sc_t in Equation 9–1), and the **Wall Damping Coefficient** (α_w in Equation 9–19) are the same as those described for the Zimont model.

18.3.3. Additional Options for the G-Equation Model

When the G-Equation model is enabled, the **G Equation Settings** group box appears. You can select either the **transport equation** or **algebraic** option for the calculation of the flame distance variance. Consult Peters Flame Speed Model in the Theory guide for the variance transport and algebraic equation expressions (Equation 9–6 and Equation 9–7). It is recommended to use the **transport equation** option for RANS and the **algebraic** option for LES.

When **Flame Curvature Source** is enabled, the curvature source term in the G-Equation, which is the last term in Equation 9–4, is included. By default, this term is excluded.

18.3.4. Defining Physical Properties for the Unburnt Mixture

The fluid material in your domain should be assigned the properties of the unburnt mixture, including the molecular heat transfer coefficient (α in Equation 9–9 in the Theory Guide), which is also referred to as the thermal diffusivity. α is defined as $k/\rho c_p$, and values at standard conditions can be found in combustion handbooks (e.g., [*42*] (p. 2211)).

For both adiabatic and non-adiabatic combustion models, you will need to specify the **Laminar Flame Speed** (U_l in Equation 9–9 in the Theory Guide) as a material property, in the **Create/Edit Materials** dialog box. When using the Zimont turbulent flame speed model, if you want to include the flame stretch effect in your model, you will also need to specify a lower value than the default **Critical Rate of Strain** (g_{cr} in Equation 9–17 in the Theory Guide). As discussed in Flame Stretch Effect in the Theory Guide, g_{cr} is set to a very high value ($1 \times 10^8 s^{-1}$) by default, so no flame stretching occurs. To include flame stretching effects, you will need to adjust the **Critical Rate of Strain** based on experimental data for the burner. Since the flame stretching and flame extinction can influence the turbulent flame speed (as discussed in Flame Stretch Effect in the Theory Guide), a realistic value for the **Critical Rate of Strain** is required for accurate predictions. Typical values for CH_4 lean premixed combustion range from 3000 to 8000 s^{-1} [*103*] (p. 2214). Note that you can specify constant values or user-defined functions to define the **Laminar Flame Speed** and **Critical Rate of Strain**. See the UDF Manual for details about user-defined functions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For adiabatic models, you will also specify the **Adiabatic Burnt Temperature** (T_{ad} in Equation 9–64 in the Theory Guide), which is the temperature of the burnt products under adiabatic conditions. This temperature will be used to determine the linear variation of temperature in an adiabatic premixed combustion calculation. You can specify a constant value or use a user-defined function.

For non-adiabatic models, you will instead specify the **Heat of Combustion** per unit mass of fuel and the **Unburnt Fuel Mass Fraction** (H_{comb} and Y_{fuel} in Equation 9–66 in the Theory Guide). ANSYS FLUENT will use these values to compute the heat losses or gains due to combustion, and include these losses/gains in the energy equation that it uses to calculate temperature. The **Heat of Combustion** can be specified only as a constant value, but you can specify a constant value or use a user-defined function for the **Unburnt Fuel Mass Fraction**.

To specify the density for a premixed combustion model, choose **premixed-combustion** in the **Density** drop-down list and set the **Adiabatic Unburnt Density** and **Adiabatic Unburnt Temperature** (T_u and ρ_u in Equation 9–67 in the Theory Guide). For adiabatic premixed models, your input for **Adiabatic Unburnt Temperature** (T_u) will also be used in Equation 9–64 in the Theory Guide to calculate the temperature.

The other properties specified for the unburnt mixture are viscosity, specific heat, thermal conductivity, and any other properties related to other models that are being used in conjunction with the premixed combustion model.

18.3.5. Setting Boundary Conditions for the Progress Variable

For premixed combustion models, you will need to set an additional boundary condition at flow inlets and exits: the progress variable, c. Valid inputs for the **Progress Variable** are as follows:

- $c=0$: unburnt mixture
- $c=1$: burnt mixture

18.3.6. Initializing the Progress Variable

Often, it is sufficient to initialize the progress variable c to 1 (burnt) everywhere and allow the unburnt ($c=0$) mixture entering the domain from the inlets to blow the flame back to the stabilizer. A better initialization is to patch an initial value of 0 (unburnt) upstream of the flame holder and a value of 1 (burnt) in the downstream region (after initializing the flow field in the **Solution Initialization** task page).

Solution Initialization → **Patch...**

See *Patching Values in Selected Cells* (p. 1242) for details about patching values of solution variables.

18.4. Setting Up the Extended Coherent Flame Model

The Extended Coherent Flame Model (ECFM) solves a transport equation for the flame surface area density, denoted Σ, in addition to the reaction progress variable. Details about setting up the reaction progress variable boundary and initial conditions, and material properties, can be found in *Setting Up the C-Equation and G-Equation Models* (p. 896). To use the ECFM, select **Extended Coherent Flame Model** as the **Premixed Model**. A list of **Extended Coherent Flame Model Constants** will appear in the **Species Model** dialog box.

For additional information, please see the following sections:

18.4.1. Modifying the ECFM Model Variant
18.4.2. Modifying the Constants for the ECFM Flame Speed Closure

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18.4.3. Setting Boundary Conditions for the ECFM Transport
18.4.4. Initializing the Flame Area Density

18.4.1. Modifying the ECFM Model Variant

The source terms for Equation 9–26 in the Theory Guide are determined by which ECFM model variant is used. There are four ECFM model variants available. The default Veynante scheme is the recommended scheme, because it provides the best accuracy in most situations. If you want access to a different model variant, you must use the following text command:

`define` → `models` → `species` → `ecfm-controls`

You can then revise the model constants as necessary. See Closure for ECFM Source Terms in the Theory Guide for further details. If you are using LES turbulence modeling, see LES and ECFM as well.

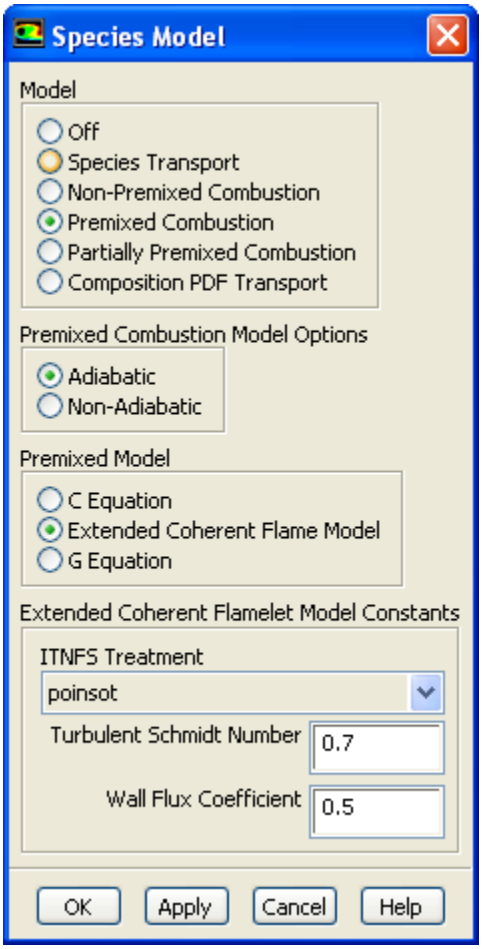
18.4.2. Modifying the Constants for the ECFM Flame Speed Closure

In the **Extended Coherent Flame Model Constants** group box of the **Species Model** dialog box, select the **ITNFS Treatment**. You have a choice of **constant-delta**, **meneveau**, **blint**, **poinsot**, or **constant**. The various ITNFS treatments are described in detail in Closure for ECFM Source Terms in the Theory Guide.

The treatment you select will determine which constants you can set. In all cases, you need to define the **Turbulent Schmidt Number** (Sc_t) and the **Wall Flux Coefficient**. If you selected the **constant-delta** treatment, you can set the **ITNFS Flame Thickness** (δ_l^0 in Equation 9–30 in the Theory Guide). If you selected the **constant** treatment, you can set the **ITNFS Value** (Γ_K in Equation 9–28 in the Theory Guide). Note that in general, you will not need to modify the constants available in the **Extended Coherent Flame Model Constants** group box, because the default values are suitable for a wide range of premixed flames.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.3 The Species Model Dialog Box for ECFM



18.4.3. Setting Boundary Conditions for the ECFM Transport

For the ECFM transport equation option for the premixed combustion models, you will need to set an additional boundary condition at flow inlets and exits: the flame area density, Σ. Valid inputs for the **Flame Area Density** are as follows:

- $\Sigma = 0$: no flame area (unburned)
- $\Sigma > 0$: burning with nonzero flame area

18.4.4. Initializing the Flame Area Density

Often, it is sufficient to initialize the flame area density Σ to 1 (laminar flame speed) everywhere and allow the unburnt mixture entering the domain from the inlets to develop. Another option for initialization is to patch an initial value of 0 (not burning) upstream of the flame holder and a value of 10 or higher (burning) in the downstream region (after initializing the flow field in the region).

See *Patching Values in Selected Cells* (p. 1242) for details about patching values of solution variables.

18.5. Postprocessing for Premixed Combustion Calculations

ANSYS FLUENT provides several additional reporting options for premixed combustion calculations. You can generate graphical plots or alphanumeric reports of the following items:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Progress Variable**
- **Damkohler Number**
- **Stretch Factor**
- **Turbulent Flame Speed**
- **Static Temperature**
- **Product Formation Rate**
- **Laminar Flame Speed**
- **Critical Strain Rate**
- **Unburnt Fuel Mass Fraction**
- **Adiabatic Flame Temperature**

These variables are contained in the **Premixed Combustion...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See *Field Function Definitions* (p. 1529) for a complete list of flow variables, field functions, and their definitions. *Displaying Graphics* (p. 1379) and *Reporting Alphanumeric Data* (p. 1509) explain how to generate graphics displays and reports of data.

Note that **Static Temperature** and **Adiabatic Flame Temperature** will appear in the **Premixed Combustion...** category only for adiabatic premixed combustion calculations; for non-adiabatic calculations, **Static Temperature** will appear in the **Temperature...** category. **Unburnt Fuel Mass Fraction** will appear only for non-adiabatic models.

ANSYS FLUENT also provides several additional reporting options for premixed combustion calculations with the ECFM model for flame speed closure. You can generate graphical plots or alphanumeric reports of the same items that are available with the premixed mode, and in addition:

- **Progress Variable Curvature**
- **Flame Area Density**
- **Net Flame Area Production**
- **Flame Area Production P1**
- **Flame Area Production P2**
- **Flame Area Production P3**
- **Flame Area Production P4**
- **Intermittent Turb Net Flame Stretch**
- **Flame Area Destruction**

These variables will also appear in the **Premixed Combustion...** category.

ANSYS FLUENT also provides two additional reporting options for premixed combustion calculations with the G-Equation model:

- **Mean Distance from Flame**
- **Variance of Distance from Flame**

The **Variance of Distance from Flame** is the variance of the distance of the instantaneous flame from the mean flame position. These variables appear in the **Premixed Combustion...** category.

For additional information, please see the following section:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18.5.1. Computing Species Concentrations

18.5.1. Computing Species Concentrations

If you know the composition of the unburnt and burnt mixtures in your model (i.e., if you have performed separate ANSYS FLUENT or external analyses of chemical equilibrium calculations or 1D premixed flames), you can compute the species concentrations in the domain using custom field functions:

- To determine the concentration of a species in the unburnt mixture, define the custom function $Y_u(1-c)$, where Y_u is the mass fraction for the species in the unburnt mixture (specified by you) and c is the value of the progress variable (computed by ANSYS FLUENT).
- To determine the concentration of a species in the burnt mixture, define the custom function $Y_b c$, where Y_b is the mass fraction for the species in the burnt mixture (specified by you) and c is the value of the progress variable (computed by ANSYS FLUENT).

See *Custom Field Functions* (p. 1579) for details about defining and using custom field functions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 19: Modeling Partially Premixed Combustion

ANSYS FLUENT provides a partially premixed combustion model that is based on the non-premixed combustion model described in *Modeling Non-Premixed Combustion* (p. 839) and the premixed combustion model described in *Modeling Premixed Combustion* (p. 893). For information about the theory behind the partially premixed combustion model, see "Partially Premixed Combustion" in the Theory Guide. Information about using the partially premixed combustion model is presented in the following sections:

19.1. Overview and Limitations
19.2. Using the Partially Premixed Combustion Model

19.1. Overview and Limitations

For additional information, please see the following sections:

19.1.1. Overview
19.1.2. Limitations

19.1.1. Overview

Partially premixed combustion systems are premixed flames with non-uniform fuel-oxidizer mixtures (equivalence ratios). Such flames include premixed jets discharging into a quiescent atmosphere, lean premixed combustors with diffusion pilot flames and/or cooling air jets, and imperfectly mixed inlets.

The partially premixed model in ANSYS FLUENT is a simple combination of the non-premixed model (*Modeling Non-Premixed Combustion* (p. 839)) and the premixed model (*Modeling Premixed Combustion* (p. 893)). The premixed reaction-progress variable, c, determines the position of the flame front. Behind the flame front ($c=1$), the mixture is burnt and the equilibrium or laminar flamelet mixture fraction solution is used. Ahead of the flame front ($c=0$), the species mass fractions, temperature, and density are calculated from the mixed but unburnt mixture fraction. Within the flame ($0<c<1$), a linear combination of the unburnt and burnt mixtures is used.

19.1.2. Limitations

The underlying theory, assumptions, and limitations of the non-premixed and premixed models apply directly to the partially premixed model. In particular, the single-mixture-fraction approach is limited to two inlet streams, which may be pure fuel, pure oxidizer, or a mixture of fuel and oxidizer. The two-mixture-fraction model extends the number of inlet streams to three, but incurs a major computational overhead. See *Limitations* (p. 894) for additional limitations.

19.2. Using the Partially Premixed Combustion Model

The procedure for setting up and solving a partially premixed combustion problem combines parts of the non-premixed combustion setup and the premixed combustion setup. An outline of the procedure is provided in *Setup and Solution Procedure* (p. 906), along with information about where to look in the non-premixed and premixed combustion chapters for details. Inputs that are specific to the partially premixed combustion model are provided in *Setup and Solution Procedure* (p. 906) and *Modifying the Unburnt Mixture Property Polynomials* (p. 907).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, please see the following sections:

19.2.1. Setup and Solution Procedure
19.2.2. Modifying the Unburnt Mixture Property Polynomials

19.2.1. Setup and Solution Procedure

1. Read your mesh file into ANSYS FLUENT and set up any other models you plan to use in conjunction with the partially premixed combustion model (turbulence, radiation, etc.).
2. Enable the partially premixed combustion model.
   a. Turn on the **Partially Premixed Combustion** model in the **Species Model** dialog box.

      Models → **Species** → **Edit...**

   b. If necessary, modify the model constants for the selected **Turbulent Flame Speed Model** in the **Species Model** dialog box. These are the same as the constants for the premixed combustion model and, in most cases, you will not need to change them from their default values. See *Modifying the Constants for the Zimont Flame Speed Model* (p. 898) for details. Similarly, you can modify the **Extended Coherent Flamelet Model Constants** if you selected the **Extended Coherent Flame Model**. See *Modifying the Constants for the ECFM Flame Speed Closure* (p. 900) for details.
3. Generate a PDF look-up table. You can follow the procedure for non-premixed combustion described in *Steps in Using the Non-Premixed Model* (p. 839).

   Important

   If ANSYS FLUENT warns you, during the partially premixed properties calculation, that any parameters are out of the range for the laminar flame speed function, you will need to modify the piecewise-linear points manually before saving the PDF file. See *Modifying the Unburnt Mixture Property Polynomials* (p. 907) for details. Also, the calculation of the thermal diffusivity uses the thermal conductivity in the **Create/Edit Materials** dialog box. More accurate thermal diffusivity polynomials can be obtained by editing the thermal conductivity in the **Create/Edit Materials** dialog box and then clicking **Recalculate Properties** in the **Premixed** tab.

4. Define the physical properties for the unburnt material in the domain.

   Materials

   ANSYS FLUENT will automatically select the **prepdf-polynomial** function for **Laminar Flame Speed**, indicating that the piecewise-linear polynomial function from the PDF look-up table will be used to compute the laminar flame speed. You may also choose to use a user-defined function instead of a piecewise-linear polynomial function. See *Defining Physical Properties for the Unburnt Mixture* (p. 898) for information about setting the other properties for the unburnt material.

5. Set the values for the mean progress variable ($\overline{c}$) and the mean mixture fraction ($\overline{f}$) and its variance ($\overline{f'^2}$) at flow inlets and exits. (For problems that include a secondary stream, you will define boundary conditions for the mean secondary partial fraction and its variance as well.)

   Boundary Conditions

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See *Defining Non-Premixed Boundary Conditions* (p. 883) for guidelines on setting mixture fraction and variance conditions, as well as thermal and velocity conditions at inlets.

Important

There are two ways to specify a premixed inlet boundary condition:

a. If you defined the fuel composition in the **Boundary** tab to be the premixed inlet species, then you should set $\overline{f}=1$ and $\overline{c}=0$ in the boundary condition dialog boxes.

b. If you set the fuel composition to pure fuel in the **Boundary** tab, you will need to set the correct equivalence ratio ($0<\overline{f}<1$) and $\overline{c}=0$ at your premixed inlet boundary condition.

For example, if the premixed inlet of methane and air is at an equivalence ratio of 0.3, you can

a. specify the mass fraction of the fuel composition of Y_{CH_4} = 0.017, Y_{O_2} = 0.236, and Y_{N_2} = 0.747 in the **Boundary** tab and $\overline{f}$ = 1 and $\overline{c}$ = 0 in the boundary condition dialog box.

b. specify the mass fraction of the fuel composition of Y_{CH_4} = 1.0 in the **Boundary** tab and $\overline{f}$ = 0.017 and $\overline{c}$ = 0 in the boundary condition dialog box.

Method (a) is preferred since it will have more points in the flame zone than method (b).

6. Initialize the value of the progress variable.

Solution Initialization → **Patch...**

See *Initializing the Progress Variable* (p. 899) for details.

7. Solve the problem and perform postprocessing.

See *Solving the Flow Problem* (p. 887) for guidelines about setting solution parameters. (These guidelines are for non-premixed combustion calculations, but they are relevant for partially premixed as well.)

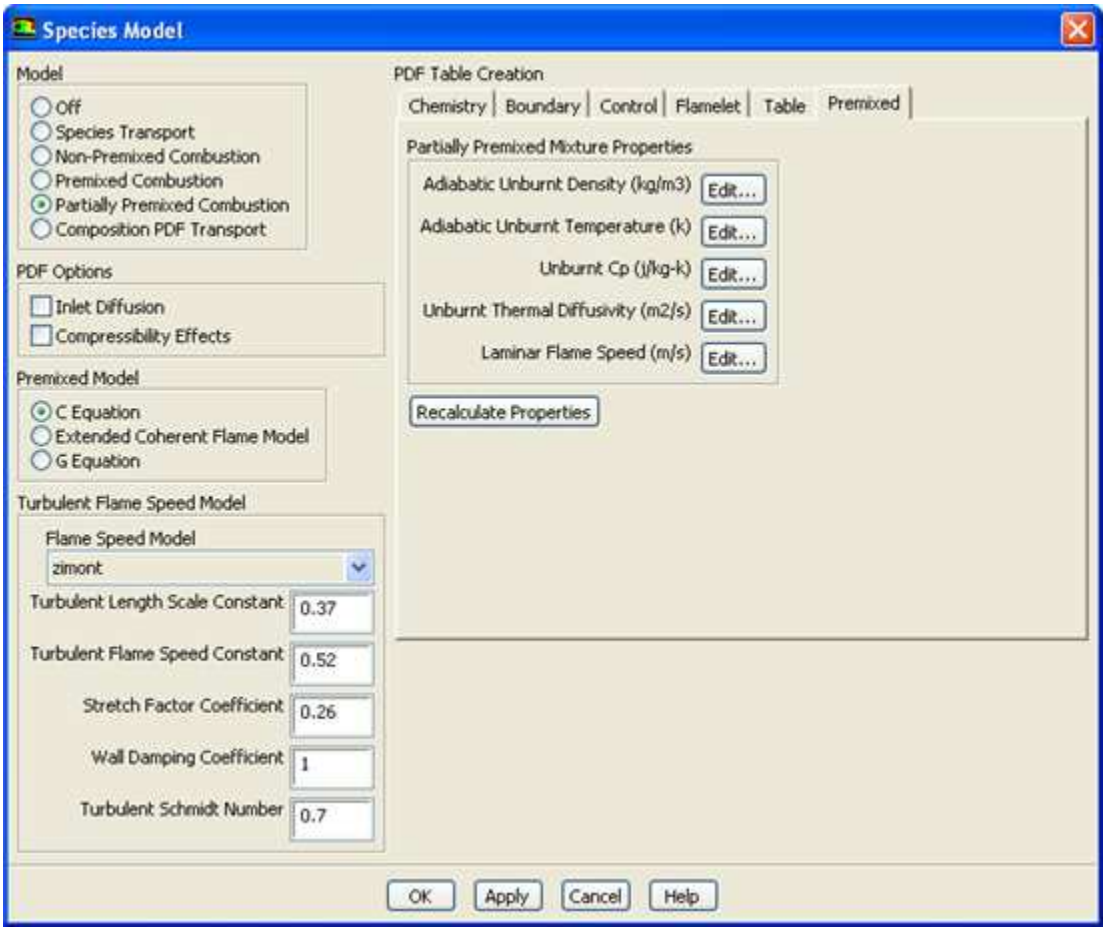
19.2.2. Modifying the Unburnt Mixture Property Polynomials

After building the PDF table, ANSYS FLUENT automatically calculates the temperature, density, heat capacity, and thermal diffusivity of the unburnt mixture as polynomial functions of the mean mixture fraction, $\overline{f}$ (see Equation 10–3 in the Theory Guide). The laminar flame speed is automatically calculated as a piecewise-linear polynomial function of $\overline{f}$.

However, as outlined in Partially Premixed Combustion Theory in the Theory Guide, the laminar flame speed depends on details of the chemical kinetics and molecular transport properties, and is not calculated directly. Instead, curve fits are made to flame speeds determined from detailed simulations [*28*] (p. 2210). These fits are limited to a range of fuels (H_2, CH_4, C_2H_2, C_2H_4, C_2H_6, and C_3H_8), air as the oxidizer, equivalence ratios of the lean limit through unity, unburnt temperatures from 298 K to 800 K, and pressures from 1 bar to 40 bars. If your parameters fall outside this range, ANSYS FLUENT will warn you when it computes the look-up table. In this case, you will need to modify the piecewise-linear points in the **Premixed** tab of the **Species Model** dialog box (*Figure 19.1* (p. 908)) before you save the PDF file.

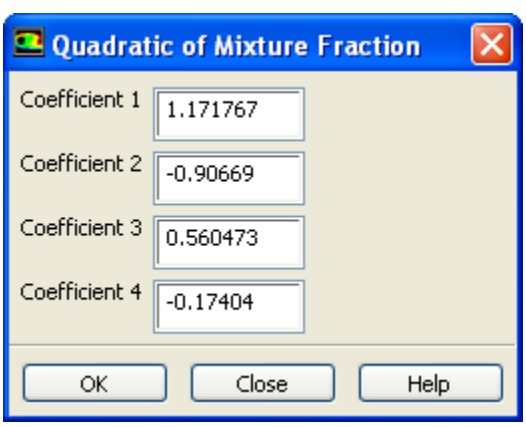
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.1 The Species Model Dialog Box (Premixed Tab)



For each polynomial function of $\overline{f}$ under **Partially Premixed Mixture Properties** (**Adiabatic Unburnt Density**, **Adiabatic Unburnt Temperature**, **Unburnt Cp**, and **Unburnt Thermal Diffusivity**), you can specify values for **Coefficient 1**, **Coefficient 2**, **Coefficient 3**, and **Coefficient 4** (the polynomial coefficients in Equation 10–3 in the Theory Guide) in the appropriate **Quadratic of Mixture Fraction** dialog box (*Figure 19.2* (p. 908)). To open this dialog box, click the appropriate **Edit...** button in the **Premixed** tab.

Figure 19.2 The Quadratic of Mixture Fraction Dialog Box

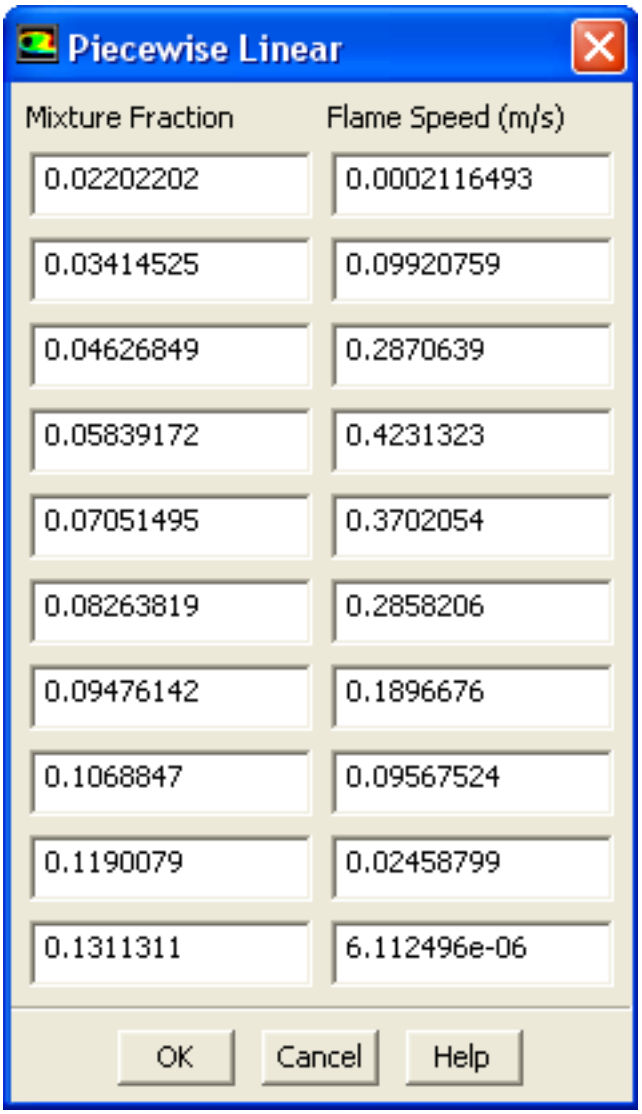


You can also specify the piecewise-linear **Mixture Fraction** and its corresponding **Laminar Flame Speed** for 10 different points in the **Piecewise Linear** dialog box (*Figure 19.3* (p. 909)). The first set of values is the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lower limit and the last set of values is the upper limit. Outside of either limit, the laminar flame speed is constant and equal to that limit. To open this dialog box, click the **Edit...** button next to **Laminar Flame Speed** in the **Premixed** tab.

Figure 19.3 The Piecewise Linear Dialog Box



Important

Note also that if you choose to use a user-defined function for the laminar flame speed in the *Create/Edit Materials Dialog Box* (p. 1736), the piecewise-linear fit becomes irrelevant.

If the secondary mixture fraction model is enabled, the unburnt properties are a function of both the mean and secondary mixture fractions. An additional column is added in the **Piecewise Linear** dialog box for the secondary mixture fraction unburnt polynomial coefficients.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 20: Modeling a Composition PDF Transport Problem

ANSYS FLUENT provides a composition PDF transport model for modeling finite-rate chemistry in turbulent flames. For information about the theory behind the composition PDF transport model, see "Composition PDF Transport" in the Theory Guide. Information about using this model is presented in the following sections:

20.1. Overview and Limitations
20.2. Steps for Using the Composition PDF Transport Model
20.3. Enabling the Lagrangian Composition PDF Transport Model
20.4. Enabling the Eulerian Composition PDF Transport Model
20.5. Initializing the Solution
20.6. Monitoring the Solution
20.7. Postprocessing for Lagrangian PDF Transport Calculations
20.8. Postprocessing for Eulerian PDF Transport Calculations

20.1. Overview and Limitations

The composition PDF transport model, like the Laminar Finite-Rate (see The Laminar Finite-Rate Model in the Theory Guide) and EDC model (see The Eddy-Dissipation-Concept (EDC) Model in the Theory Guidee), should be used when you are interested in simulating finite-rate chemical kinetic effects in turbulent reacting flows. With an appropriate chemical mechanism, kinetically-controlled species such as CO and NOx, as well as flame extinction and ignition, can be predicted. PDF transport simulations are computationally expensive, and it is recommended that you start your modeling with small meshes, and preferably in 2D.

A limitation that applies to the composition PDF transport model is that you must use the pressure-based solver as the model is not available with the density-based solver.

ANSYS FLUENT has two different discretizations of the composition PDF transport equation, namely Lagrangian and Eulerian. The Lagrangian method is strictly more accurate than the Eulerian method, but requires significantly longer run time to converge.

20.2. Steps for Using the Composition PDF Transport Model

The procedure for setting up and solving a composition PDF transport problem is outlined below, and then described in detail. Remember that only steps that are pertinent to composition PDF transport modeling are shown here. For information about inputs related to other models that you are using in conjunction with the composition PDF transport model, see the appropriate sections for those models.

1. Read a CHEMKIN-formatted gas-phase mechanism file and the associated thermodynamic data file in the **CHEMKIN Mechanism** dialog box (see *Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827)).

   File → **Import** → **CHEMKIN Mechanism...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If your chemical mechanism is not in CHEMKIN format, you will have to enter the mechanism into ANSYS FLUENT as described in *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799).

2. Enable a turbulence model.

   Models → **Viscous** → **Edit...**

3. Enable the **Composition PDF Transport** model and set the related parameters. Refer to *Enabling the Lagrangian Composition PDF Transport Model* (p. 913) and *Enabling the Eulerian Composition PDF Transport Model* (p. 915) for further details.

   Models → **Species** → **Edit...**

4. Check the material properties in the **Create/Edit Materials** dialog box and the reaction parameters in the **Reactions** dialog box. The default settings should be sufficient.

   Materials

5. Set the operating conditions, cell zone conditions, and boundary conditions.

   Cell Zone Conditions → **Operating Conditions...**

   Boundary Conditions

6. Check the solver settings.

   Solution Methods

   Solution Controls

   The default settings should be sufficient, although it is recommended to change the discretization to second-order once the solution has converged.

7. Initialize the solution. You may need to patch a high-temperature region to ignite the flame.

   Solution Initialization → **Initialize**

   Solution Initialization → **Patch...**

8. Run the solution.

   Run Calculation

9. Solve the problem and perform postprocessing.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

A good initial condition can reduce the solution time substantially. It is recommended to start from an existing solution calculated using the Laminar Finite-Rate, EDC model, non-premixed combustion model, or partially premixed combustion model. See *Modeling Species Transport and Finite-Rate Chemistry* (p. 799), *Modeling Non-Premixed Combustion* (p. 839), and *Modeling Partially Premixed Combustion* (p. 905) for further information on such simulations.

20.3. Enabling the Lagrangian Composition PDF Transport Model

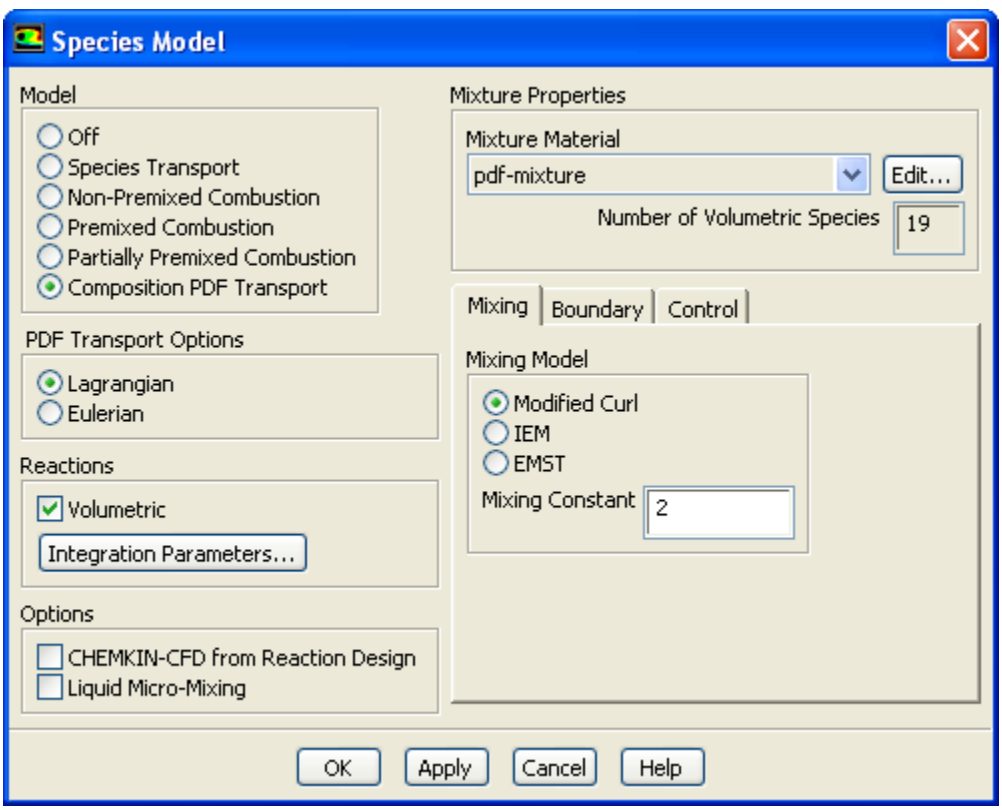
To enable the composition PDF transport model, select **Composition PDF Transport** in the **Species Model** dialog box (*Figure 20.1* (p. 913)).

Models → **Species** → **Edit...**

When you enable **Composition PDF Transport**, the dialog box will expand to show the relevant inputs.

1. Select **Lagrangian** under **PDF Transport Options**.

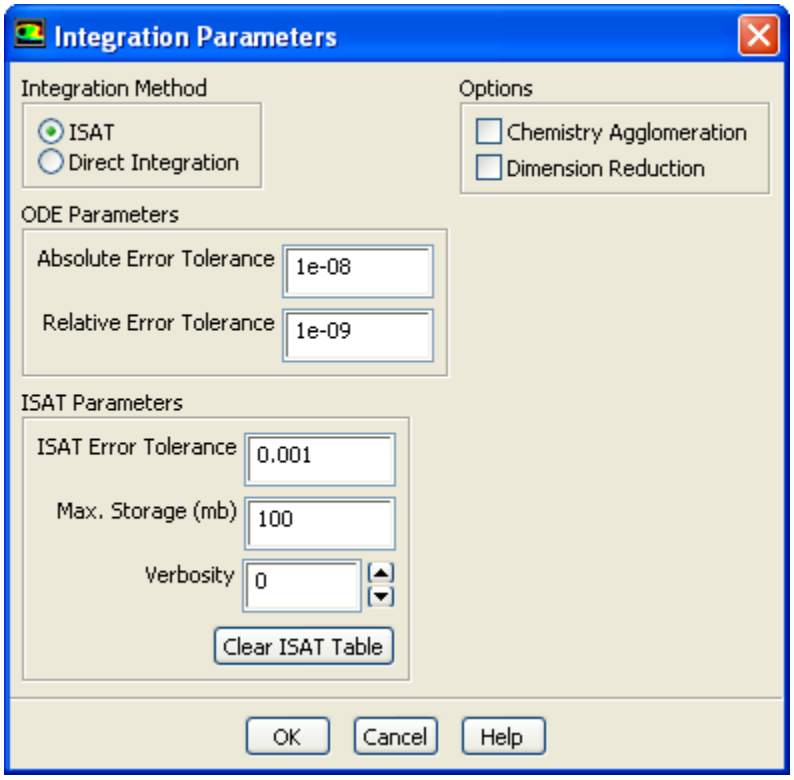
Figure 20.1 The Species Model Dialog Box for Lagrangian Composition PDF Transport



2. Enable **Volumetric** under **Reactions**.
3. Click the **Integration Parameters...** button to open the *Integration Parameters Dialog Box* (p. 1694) (*Figure 20.2* (p. 914)). For additional information, see *Using ISAT* (p. 924).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.2 The Integration Parameters Dialog Box



4. Enable **CHEMKIN-CFD from Reaction Design** to allow you to use reaction rates from Reaction Design's CHEMKIN module, instead of the default ANSYS FLUENT reaction rates. ANSYS FLUENT's ISAT algorithm is employed to integrate these rates. Please refer to the manual [2] (p. 2209) from Reaction Design for details on the chemistry formulation options. For more information, or to obtain a license to use the FLUENT/CHEMKINmodule, please contact Reaction Design at `info@reactiondesign.com` or +1 858-550-1920, or go to `www.reactiondesign.com`.

5. Enable **Liquid Micro-Mixing** to interpolate C_ϕ from turbulence models and scalar spectra, as noted in Liquid Reactions in the Theory Guidee. This is applied to cases where reactions in liquids occur at low turbulence levels, among reactants with low diffusivities. Therefore, a default value of $C_\phi = 2$ may not be desirable, as it over-estimates the mixing rate.

6. In the **Mixing** tab, select **Modified Curl**, **IEM**, or **EMST** under **Mixing Model** and specify the value of the **Mixing Constant** (C_ϕ in Equation 11–6 in the Theory Guide). For more information about particle diffusion, see Particle Mixing in the Theory Guide.

Important

If the **Liquid Micro-Mixing** option is enabled, you cannot set the **Mixing Constant**.

7. You will not be specifying species boundary conditions in the **Boundary** tab. This is only applicable to the **Eulerian PDF Transport Option**.

8. In the **Control** tab, you will specify the Lagrangian PDF transport control parameters.

Particles Per Cell

sets the number of PDF particles per cell. Higher values of this parameter will reduce statistical error, but increase computational time.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local Time Stepping

is available for steady-state simulations and can increase the convergence rate by taking maximum allowable time-steps on a cell-by-cell basis. (see Equation 11–4 of the Theory Guide). If **Local Time Stepping** is enabled, then you can specify the following parameters:

Convection #

specifies the particle convection number (see Δt_{conv} in Equation 11–4 in the Theory Guide).

Diffusion #

specifies the particle diffusion number (see Δt_{diff} in Equation 11–4).

Mixing #

specifies the particle mixing number (see Δt_{mix} in Equation 11–4).

20.4. Enabling the Eulerian Composition PDF Transport Model

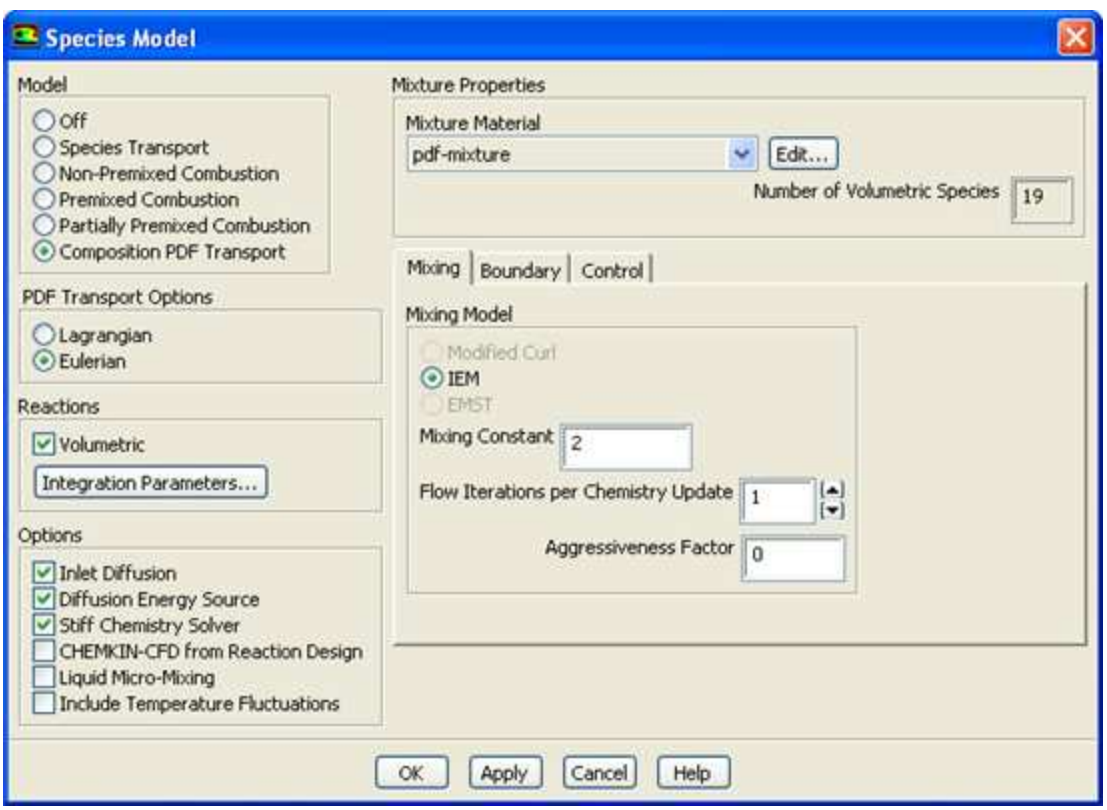
To enable the composition PDF transport model, select **Composition PDF Transport** in the **Species Model** dialog box (*Figure 20.1* (p. 913)).

Models → **Species** → **Edit...**

When you enable **Composition PDF Transport**, the dialog box will expand to show the relevant inputs.

1. Select **Eulerian** under **PDF Transport Options**.

Figure 20.3 The Species Model Dialog Box for Eulerian Composition PDF Transport



2. Enable **Volumetric** under **Reactions**. The **Stiff Chemistry Solver** is disabled by default and should be enabled if the kinetic mechanism is numerically stiff.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Integration Parameters...** button to open the *Integration Parameters Dialog Box* (p. 1694) (*Figure 20.2* (p. 914)). See *Using ISAT* (p. 924) for detailed information about this dialog box.
4. Enable **Inlet Diffusion** to include the diffusive transport of species through the inlets of your domain. Disable this option if the convective flux at one of the inlets is very small, resulting in mass loss by diffusion through that inlet.
5. Make sure that **Diffusion Energy Source** is enabled if you want to include species diffusion effects in the energy equation.
6. Enable **Liquid Micro-Mixing** if the fuel and oxidizer are liquids with low diffusivities (high Schmidt numbers). In this case, The **Mixing Constant** will be calculated per Equation 11–13 in the Theory Guide.
7. By default, the Laminar (one mode) energy equation is solved, where temperature fluctuations are ignored. By enabling **Include Temperature Fluctuations**, the multi-mode energy equation will be solved as done for species.
8. In the **Mixing** tab, only the **IEM** mixing model is available for Eulerian PDF transport.
   a. Specify the value of the **Mixing Constant**. The default value is 2 for gas phase species.

      Important

      If the **Liquid Micro-Mixing** option is enabled, you cannot set the **Mixing Constant**.

   b. Enter the number of **Flow Iterations per Chemistry Update**. This is the frequency at which ANSYS FLUENT will update the chemistry during the calculation. Increasing the number can reduce the computational expense of the chemistry calculations.
   c. Enter the **Aggressiveness Factor**. This is a numerical factor which controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 (the default) is the most robust, but results in the slowest convergence.
9. In the **Boundary** tab, define the compositions of the fuel and oxidizer.

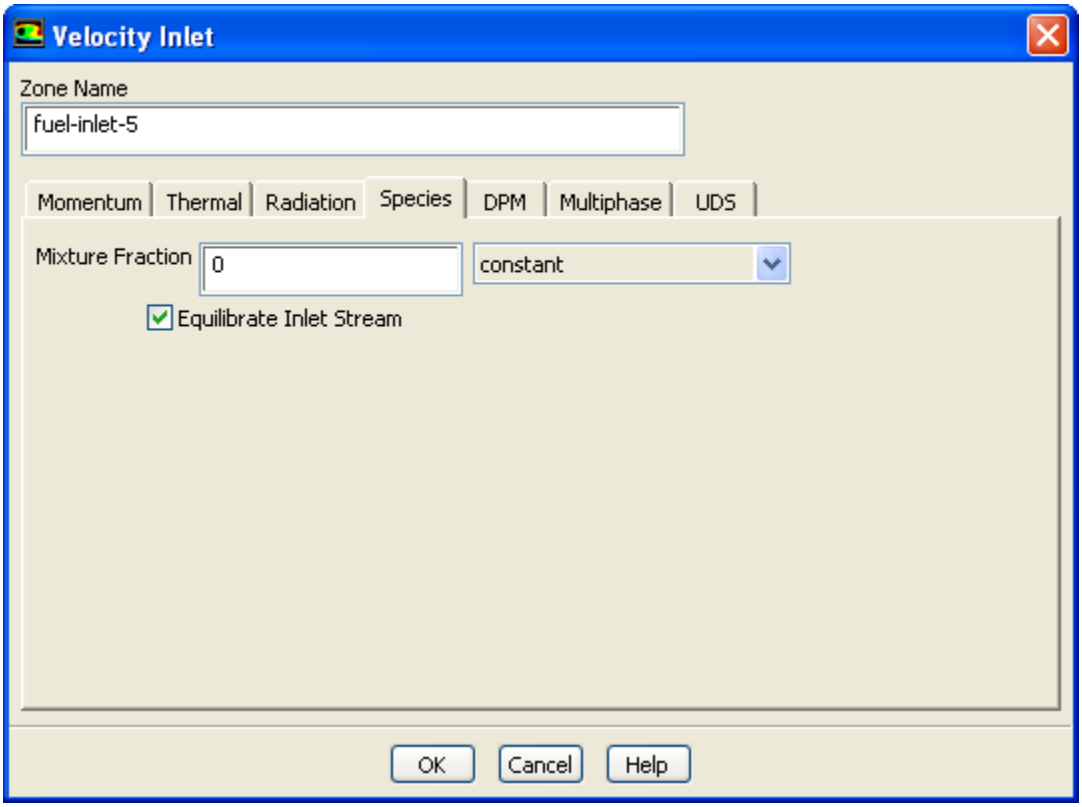
For additional information, please see the following sections:
20.4.1. Defining Species Boundary Conditions

20.4.1. Defining Species Boundary Conditions

At flow inlets, specify the **Mixture Fraction** in the **Species** tab of the boundary condition inlet dialog boxes, as shown in *Figure 20.4* (p. 917). At outlet boundaries, similarly specify the **Backflow Mixture Fraction**. ANSYS FLUENT always applies zero flux boundary conditions at walls for all species.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.4 The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport



20.4.1.1. Equilibrating Inlet Streams

The **Equilibrate Inlet Stream** option in the **Species** tab of the boundary conditions dialog boxes will set the inlet compositions to their chemical equilibrium values. This option can be used to model pilot flames and exhaust gas recirculation.

Important

This option should not be enabled for pure fuel or oxidizer inlets.

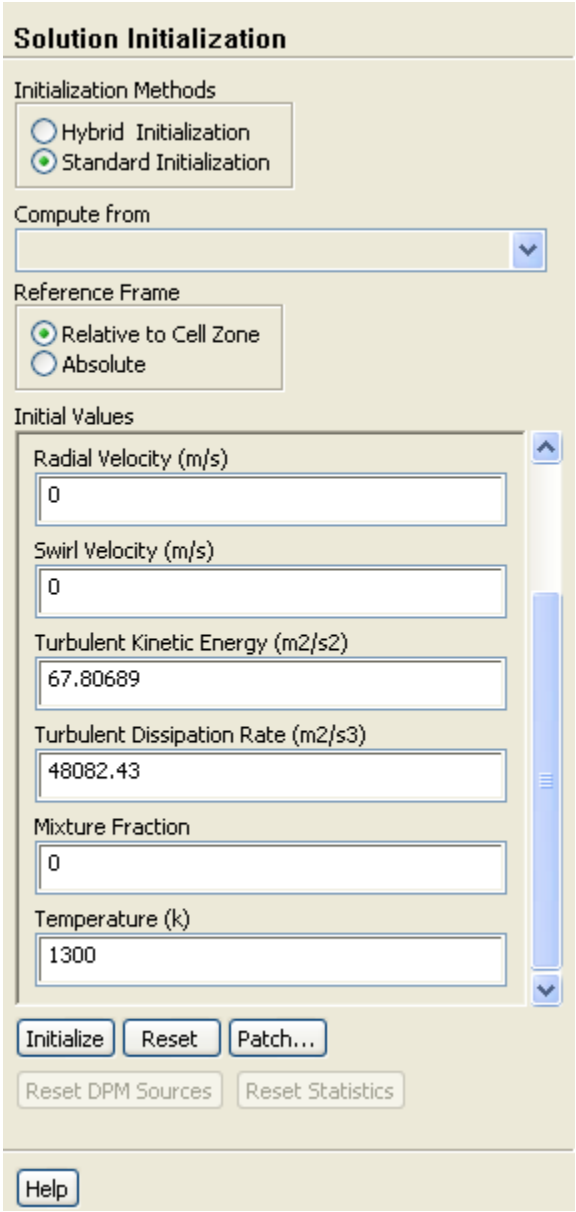
If you are using the Eulerian PDF transport model, specify the discretization and under-relaxation for the **Eulerian PDF** composition transport in the **Solution Controls** and **Solution Methods** task page.

20.5. Initializing the Solution

For the Eulerian PDF Transport model, the initialization variables are the mixture fraction and the temperature. The species mass fractions and enthalpy are calculated based on the fuel and oxidizer composition (specified in the **Species** dialog box) and the initialized mixture fraction and temperature.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.5 The Solution Initialization Task Page for Eulerian Composition PDF Transport

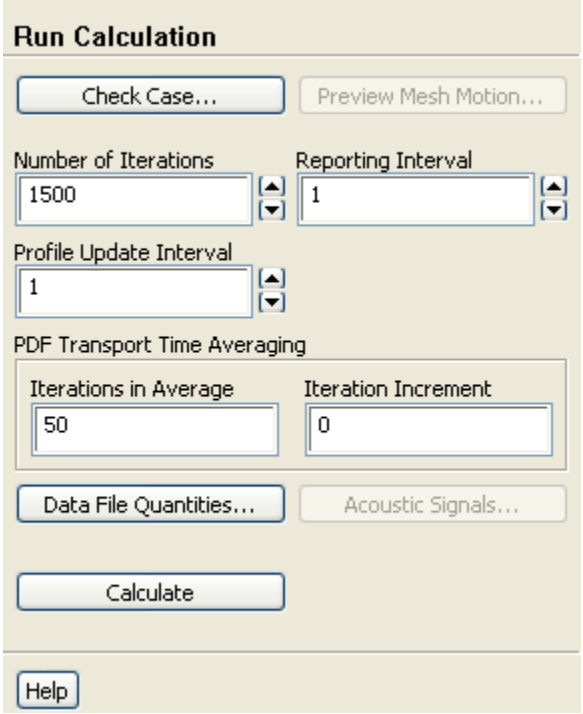


20.6. Monitoring the Solution

At low speeds, combustion couples to the fluid flow through density. The Lagrangian PDF transport algorithm has random fluctuations in the density field, which in turn causes fluctuations in the flow field. For steady-state flows, statistical fluctuations are decreased by averaging over a number of previous iterations in the **Run Calculation** task page (*Figure 20.6* (p. 919)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.6 The Run Calculation Task Page for Composition PDF Transport



Averaging reduces statistical fluctuations and stabilizes the solution. However, ANSYS FLUENT often indicates convergence of the flow field before the composition fields (temperatures and species) are converged. You should lower the default convergence criteria in the *Residual Monitors Dialog Box* (p. 1915), and always check that the **Total Heat Transfer Rate** in the *Flux Reports Dialog Box* (p. 2027) is balanced. It is also recommended that you monitor temperature/species on outlet boundaries and ensure that these are steady.

The Lagrangian PDF method has the additional solution controls: **Iterations in Average** and **Iteration Increment**. By increasing the **Iterations in Average**, fluctuations are smoothed out and residuals level off at smaller values. However, the composition PDF method requires a larger number of iterations to reach steady-state. It is recommended that you use the default of 50 **Iterations in Average** until the steady-state solution is obtained. Then, to gradually decrease the residuals, increase the **Iterations in Average** by setting a **Iteration Increment** to a value from 0 to 1 (the value 0.2 is recommended). Subsequent iterations will increase the **Iterations in Average** by the **Iteration Increment**.

For additional information, please see the following sections:

20.6.1. Running Unsteady Composition PDF Transport Simulations
20.6.2. Running Compressible Lagrangian PDF Transport Simulations
20.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer

20.6.1. Running Unsteady Composition PDF Transport Simulations

For unsteady Lagrangian composition PDF transport simulations, a fractional step scheme is employed where the PDF particles are advanced over the time step, and then the flow is advanced over the time step. Unlike steady-state simulations, composition statistics are not averaged over iterations, and to reduce statistical error you should increase the number of particles per cell in the **Solution Monitors** dialog box.

For low speed flows, the thermo-chemistry couples to the flow through density. Statistical errors in the calculation of density may cause convergence difficulties between time step iterations. If you experience this, increase the number of PDF particles per cell, or decrease the density under-relaxation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.6.2. Running Compressible Lagrangian PDF Transport Simulations

Compressibility is included when **ideal-gas** is selected as the density method in the *Create/Edit Materials Dialog Box* (p. 1736). For such flows, particle internal energy is increased by $p\Delta v$ over the time step Δt, where p is the cell pressure and Δv is the change in the particle specific volume over the time step.

20.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer

When solid zones are present in the simulation, ANSYS FLUENT solves the energy equation in the turbulent flow zones by the Lagrangian Monte Carlo particle method, and the energy equation in the solid zones by the finite-volume method.

20.7. Postprocessing for Lagrangian PDF Transport Calculations

For additional information, please see the following sections:

20.7.1. Reporting Options
20.7.2. Particle Tracking Options

20.7.1. Reporting Options

ANSYS FLUENT provides several reporting options for the Lagrangian composition PDF transport calculations. You can generate graphical plots or alphanumeric reports of the following items:

- **Static Temperature**
- **Mean Static Temperature**
- **RMS Static Temperature**
- **Mass fraction of species-n**
- **Mean species-n Mass Fraction**
- **RMS species-n Mass Fraction**

The instantaneous composition (**Static Temperature** and **Mass fraction of species-n**) in a cell are calculated as,

$$\phi_{instant} = \frac{\sum_{i=1}^{N_c} \phi_p m_p}{\sum_{i=1}^{N_c} m_p} \qquad \textbf{(20–1)}$$

where

$\phi_{instant}$ = instantaneous cell species mass fraction or temperature at the present iteration

N_c = number of particles in the cell

T_p = particle mass fraction or temperature

m_p = particle mass

Mean and root-mean-square (RMS) temperatures are calculated in ANSYS FLUENT by averaging instantaneous temperatures over a user-specified number of previous iterations (see *Monitoring the Solution* (p. 918)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

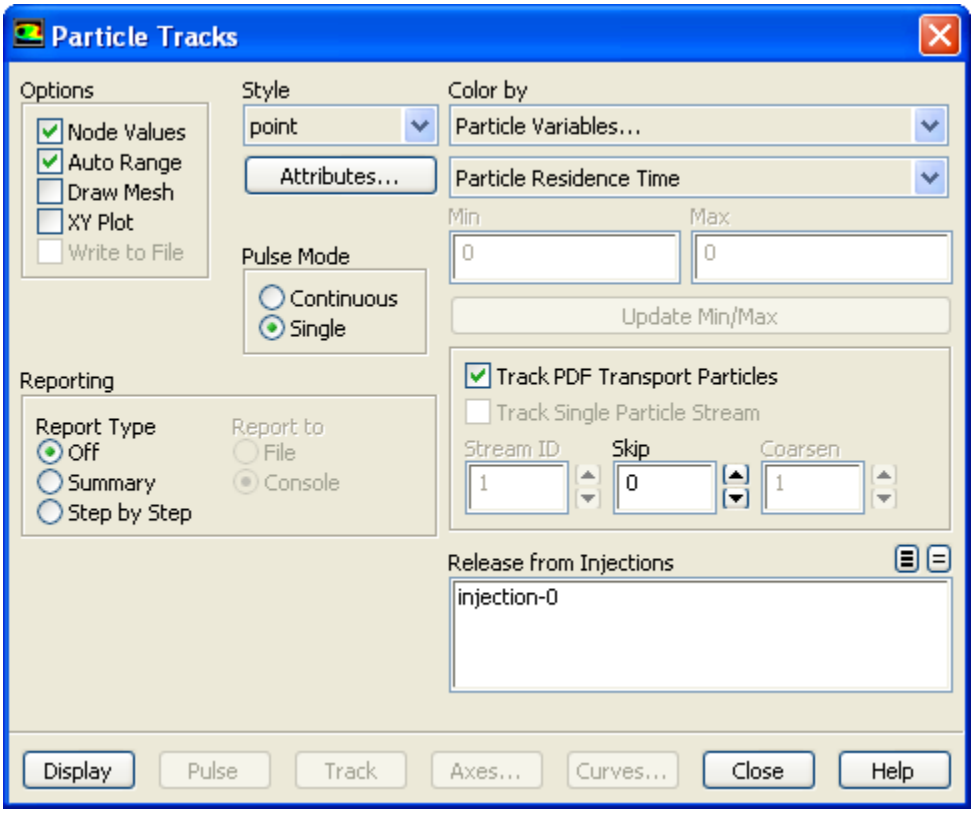
Note that for steady-state simulations, instantaneous temperatures and species represent a Monte Carlo realization and are as such unphysical. Mean and RMS quantities are much more useful.

20.7.2. Particle Tracking Options

When you have enabled the Lagrangian composition PDF transport model, you can display the trajectories of the PDF particles using the *Particle Tracks Dialog Box* (p. 1980) (*Figure 20.7* (p. 921)).

Graphics and Animation → **Particle Tracks** → **Set Up...**

Figure 20.7 The Particle Tracks Dialog Box for Tracking PDF Particles



Select the **Track PDF Transport Particles** option to enable PDF particle tracking. To speed up the plotting process, you can specify a value n for **Skip**, which will plot only every nth particle. For details about setting other parameters in the **Particle Tracks** dialog box, see *Displaying of Trajectories* (p. 1068).

When you have finished setting parameters, click **Display** to display the particle trajectories in the graphics window.

20.8. Postprocessing for Eulerian PDF Transport Calculations

For additional information, please see the following sections:

20.8.1. Reporting Options

20.8.1. Reporting Options

To postprocess the Eulerian composition PDF transport model, the following variables are available for each mode:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mixture fraction**
- **Mass fraction of species in mode n**
- **Temperature of mode n**
- **Sensible Enthalpy of mode n**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 21: Using Chemistry Acceleration

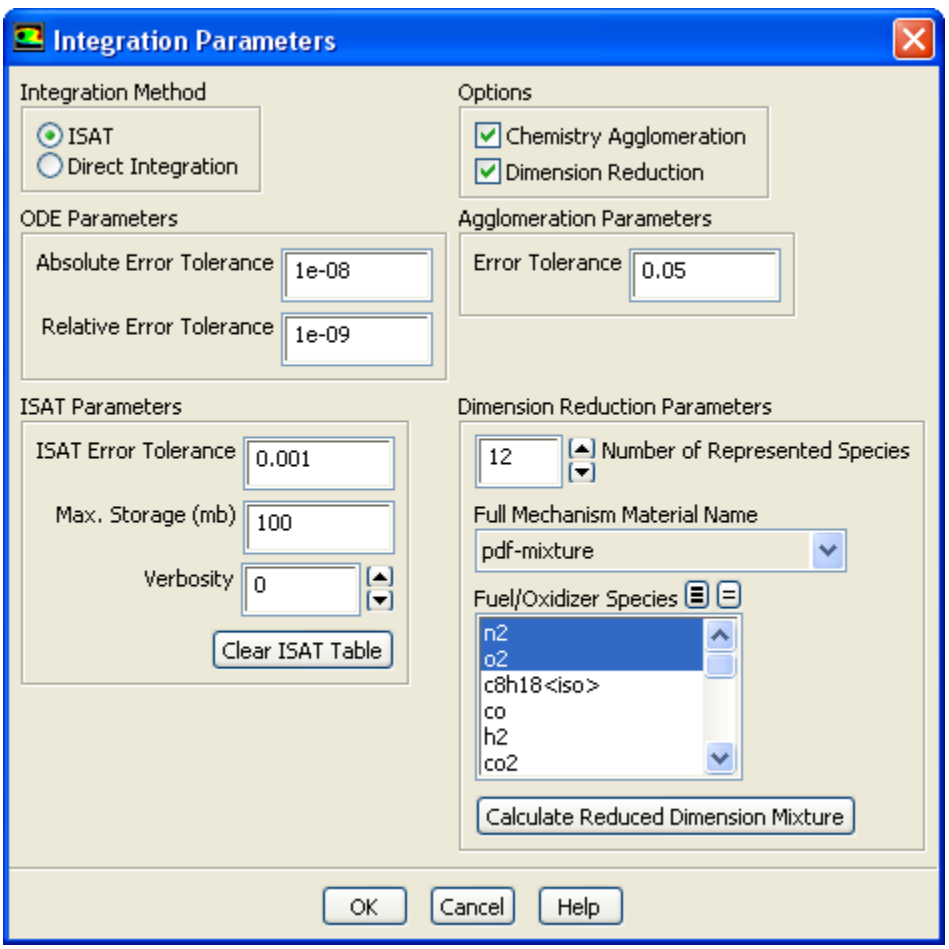
ANSYS FLUENT can model detailed chemical kinetics in laminar and turbulent flames. Laminar flames are modeled with the **Laminar Finite-Rate** option, while four turbulence-chemistry interaction models are available for turbulent flames (**Laminar Finite-Rate**, **Eddy-Dissipation Concept**, **Lagrangian PDF Transport**, and **Eulerian PDF Transport**). Detailed chemical mechanisms are invariably numerically stiff and compute-intensive. ANSYS FLUENT provides three methods to accelerate these computations:

- ISAT
- Chemistry Agglomeration
- Dimension Reduction

All three methods are enabled or disabled in the **Integration Parameters** dialog box, accessed from the **Species Model** dialog box:

Models → **Species** → **Edit...**

Figure 21.1 The Integration Parameters Dialog Box Displaying Dimension Reduction



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All three acceleration methods induce some accuracy loss, and the controlling parameters should be carefully adjusted to ensure that this inaccuracy is acceptable, see "Chemistry Acceleration". Further information about using the three chemistry acceleration models is presented in the following sections:

21.1. Using ISAT
21.2. Using Chemistry Agglomeration
21.3. Dimension Reduction

21.1. Using ISAT

In-Situ Adaptive Tabulation (ISAT) is a storage-retrieval method that constructs a chemistry table at run-time (in-situ) with a user-specified interpolation accuracy (adaptive tabulation). ANSYS FLUENT uses an ODE solver to integrate the stiff chemical kinetics.

The stiff ODE integrator has two error tolerances: the **Absolute Error Tolerance** and the **Relative Error Tolerance** under **ODE Parameters**, that are set to default values of 10^{-8} and 10^{-9} respectively. These should be sufficient for most applications, although these tolerances may need to be decreased for some cases such as ignition. For problems in which the accuracy of the chemistry integrations is crucial, it may be useful to test the accuracy of the error tolerances in simple zero-dimensional and one-dimensional test simulations with parameters comparable to those in the full simulation.

For additional information, please see the following sections:

21.1.1. ISAT Parameters
21.1.2. Monitoring ISAT
21.1.3. Using ISAT Efficiently
21.1.4. Reading and Writing ISAT Tables

21.1.1. ISAT Parameters

If you have selected **ISAT** under **Integration Method**, you will then be able to set additional ISAT parameters. The numerical error in the ISAT table is controlled by the **ISAT Error Tolerance** under **ISAT Parameters**. The default **ISAT Error Tolerance** of 0.001 may be sufficiently accurate for temperature and major species, but will most likely need to be decreased to get accurate minor species and pollutant predictions. For steady-state simulations, it may help to start with a high error tolerance during the initial iterations towards a converged solution. A larger error tolerance results in smaller tables and quicker run times, but greater error.

Important

After your steady simulation is converged, you should always decrease the **ISAT Error Tolerance** and perform further iterations until the species that you are interested in are unchanged.

The **Max. Storage** is the maximum RAM used by the ISAT table, and has a default value of 100 MB. It is recommended that you set this parameter to a large fraction of the available memory on your computer. The value of **Verbosity** allows you to monitor ISAT performance in different levels of detail. See *Monitoring ISAT* (p. 925) for details about this parameter.

Chemistry agglomeration is available for all models that use ISAT. Enable **Chemistry Agglomeration** in the **Integration Parameters** dialog box. The chemistry agglomeration **Error Tolerance** (ε_{tol}) can be set in the **Integration Parameters** dialog box. Larger values of ε_{tol} result in a larger number of cells agglomerated, fewer calls to the reaction integrator, increased run-time speed, but greater error. More information can be found in Chemistry Agglomeration in the Theory Guide.

To purge the ISAT table, click the **Clear ISAT Table** button. See *Using ISAT Efficiently* (p. 925) for more details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21.1.2. Monitoring ISAT

You can monitor ISAT performance by setting the **Verbosity** in the **Integration Parameters** dialog box. For a **Verbosity** of 1 or 2, ANSYS FLUENT writes the following information periodically to a file named `case-file-name_stats.out`:

- total number of *queries*
- total number of queries resulting in *retrieves*
- total number of queries resulting in *grows*
- total number of queries resulting in *adds*
- total number of queries resulting in *direct integrations*
- cumulative CPU seconds in ISAT
- cumulative CPU seconds outside ISAT
- cumulative wall-clock time in seconds (i.e., total CPU time in ISAT plus total CPU time out of ISAT plus CPU idle time)

The ISAT **Verbosity** option of 2 is for expert users who are familiar with ISATAB v5.0 [*68*] (p. 2212). ANSYS FLUENT writes out the following files for **Verbosity** = 2:

- `tablename_stats.out`, as described above
- `tablename_ODE_accuracy.out` reports the accuracy of the ODE integrations. For every new ISAT table entry, if the maximum absolute error in temperature or species is greater than any previous error, a line is written to this file. This line consists of the total number of ODE integrations performed up to this time, the maximum absolute species error, the absolute temperature error, the initial temperature and the time step.
- `tablename_ODE_diagnostic.out` prints diagnostics from the ODE solver
- `tablename_ODE_warning.out` prints warnings from the ODE solver

Initially, the table name is equal by default to the current case name, and is changed as the table is written or read.

In parallel, each processor builds its own ISAT table. If **Verbosity** is enabled in parallel, each compute node writes out the **Verbosity** file(s) with the node ID number appended to the file name.

21.1.3. Using ISAT Efficiently

Efficient use of ISAT requires thoughtful control. What follows are some detailed recommendations concerning the achievement of this goal.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The numerical error in the ISAT table is controlled by the **ISAT Error Tolerance**, which has a default value of 0.001. This value is relatively large, which allows faster convergence times for steady-state simulations. However, once the solution has converged, it is important to reduce this **ISAT Error Tolerance** and re-converge. This process should be repeated until the species that you are interested in modeling are unchanged. Unsteady simulations should be run with a sufficiently small **ISAT Error Tolerance** so that the species of interest are unaffected by this parameter. Note that as the error tolerance is decreased, the memory and time requirements to build the ISAT table will increase substantially. There is a large performance penalty in specifying an error tolerance smaller than is needed to achieve acceptable accuracy, and the error tolerance should be decreased gradually and judiciously.

Important

Once the ISAT table is full, all queries that cannot be retrieved are directly integrated. Since retrieves are much quicker than direct integrations, larger ISAT tables are faster. Hence, you should set the ISAT **Max. Storage** to a large fraction of the available memory on your computer.

During the initial iterations, before a steady-state solution is attained, transient composition states occur that are not present in the steady-state solution. For example, you might patch a high temperature region in a cold fuel-air mixing zone to ignite the flame, whereas the converged solution never has hot reactants without products. Since all states that are realized in the simulation are tabulated in ISAT, these initial mappings are wasteful of memory, and can degrade ISAT performance. If the table fills the allocated memory and contains entries from an initial transient that are no longer accessed, it may be beneficial to purge the ISAT table. This is achieved by either clearing it in the **Integration Parameters** dialog box, or saving your case and data files, exiting ANSYS FLUENT, then restarting ANSYS FLUENT and reading in the case and data.

From experience, ISAT performs very well on premixed turbulent flames, where the range of composition states are smaller than in non-premixed flames. ISAT performance degrades in flames with large residence times, where more work is required in the ODE integrator.

21.1.4. Reading and Writing ISAT Tables

ANSYS FLUENT can write and subsequently read ISAT tables. However, it is in general not recommended to write and read ISAT tables for the following reason: ISAT tabulates chemical states that are specific to a single simulation. The ISAT tabulated composition states are determined by the geometry, boundary conditions, physics models such as turbulence and radiation, thermodynamic and transport properties, as well the chemical mechanism. If any of these parameters change, the realized composition space changes, and significant parts of the existing ISAT table are no longer accessed. These un-accessed ISAT table entries slow interpolation and decrease the number of useful accessed table entries that can be added. Since the time consumed building the ISAT table (hours) is typically much smaller than simulation run-times (days), it is advised to re-build the table when restarting.

When ANSYS FLUENT is run in parallel, each partition builds its own ISAT table and does not exchange information with ISAT tables on other compute nodes. You can save the ISAT tables on all compute nodes:

File → **Write** → **ISAT Table...**

Each compute node writes out its ISAT table to the specified file name, with the node ID number appended to the file name. For example, a specified file name of `my_name` on a two compute node run will write two files called `my_name-0.isat` and `my_name-1.isat`.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subsequent runs can start from existing ISAT tables by reading them into memory.

File → **Read** → **ISAT Table...**

Files can be read in two ways:

- Parallel nodes can read in corresponding ISAT tables saved from a previous parallel simulation. The appended node ID should not be removed from the input file name.

  Note

  You should never read ISAT tables generated from a parallel simulation with a different number of parallel nodes.

- All nodes can read one unique ISAT table. You might use this approach if you have a large table from a serial simulation. ANSYS FLUENT first checks to see if the exact file name that you specified exists, and if it does, all nodes will read this one file.

21.2. Using Chemistry Agglomeration

The Chemistry Agglomeration method reduces the number of calls to the computationally expensive ODE integrator by clustering cells with similar compositions. The size of these clusters is determined by the **Agglomeration Parameters Error Tolerance** (ε_{tol}) in the **Integration Parameters** dialog box. Larger values of ε_{tol} result in a larger number of agglomerated cells, fewer calls to the reaction integrator, increased run-time speed, but greater error. More information can be found in Chemistry Agglomeration in the Theory Guide.

Important

The default **Error Tolerance** of 0.05 is relatively large. This enables faster convergence times for steady-state simulations. However, once the solution has converged, it is important to reduce this agglomeration **Error Tolerance** and re-converge. This process should be repeated until the species that you are interested in modeling are unchanged. Unsteady simulations should be run with a sufficiently small **Error Tolerance** so that the species of interest are unaffected by this parameter. However, since performance can decrease substantially when the **Error Tolerance** is decreased, care should be taken to ensure that this parameter is not smaller than is needed to achieve acceptable accuracy.

21.3. Dimension Reduction

Detailed kinetic mechanisms typically contain a multitude of intermediate species that far exceed the number of major fuel, oxidizer, and product species. Chemical mechanism **Dimension Reduction** reduces the number of intermediate species transport equations (called representative species) that are solved, and reconstructs the 'unrepresented' species using chemical equilibrium assumptions.

Important

Since ANSYS FLUENT is limited to a maximum of 50 transported species, the main use of **Dimension Reduction** is to enable simulation with chemical mechanisms containing more than 50 species.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Follow these steps to use **Dimension Reduction**:

- Import your CHEMKIN mechanism. Note that you can import CHEMKIN mechanisms which contain more than 50 species.
- Click the **Integration Parameters** button in the **Species Model** dialog box, and enable **Dimension Reduction**.
- Set the **Number of Represented Species**. This must be greater than 10 and less than the number of species in the full mechanism. The **Number of Represented Species** must also be less than 50 minus the number of unrepresented elements (the number of chemical elements in the unrepresented species). A larger **Number of Represented Species** will increase accuracy, but also increase computational expense. The default of 12 has been chosen to provide a good compromise between accuracy and speed.
- Select the **Full Mechanism Material Name**, which is typically the name of the CHEMKIN mechanism that you imported. ANSYS FLUENT can store several imported CHEMKIN mechanisms in different mixture materials whose mechanisms you are not using. However, since mechanisms are typically large, it is recommended to delete unused mixtures to reduce memory requirements.
- Set the boundary and initial fuel and oxidizer, as well as product species, as represented species. These are input in the **Fuel/Oxidizer Species** list. Note that you can force other species to be represented by selecting them here. Species of interest, especially species that are not near chemical equilibrium, such as pollutants, and their associated intermediate species in the mechanism should also be included. Intermediate species that occur in large mass fractions relative to the fuel and oxidizer species should be included, as well as species important in the chemical pathway. For example, for methane combustion in air, CH_3 should be included as a represented species since CH_4 pyrolizes to CH_3 first.
- Click **Calculate Reduced Dimension Mixture**. This will create a new mixture material called `reduced-dimension-mixture`, which contains the represented species as well as proxy 'species' for the unrepresented elements. These unrepresented elements have "u" prepended to the element name. You should never rename the reduced-dimension-mixture mixture material, or select another mixture as the active material, while **Dimension Reduction** is enabled.
- Continue with the setup, solution, and postprocessing as for other detailed chemistry cases. Note that the boundary and initial mass fraction of unrepresented elements should always be zero. The entries for the unrepresented element mass fractions are disabled in the ANSYS FLUENT GUI.

For postprocessing, the unrepresented elements are available in the **Species** list as the element name with "u" prepended. All species in the full mechanism, consisting of both represented and unrepresented species, are available in the **Full Mechanism Species...** category of the postprocessing dialog boxes.

After you obtain a preliminary solution with **Dimension Reduction**, it is recommended that you check the magnitude of all unrepresented species, which are available in the **Full Mechanism Species...** option in the **Contours** dialog box. If the mass fraction of any unrepresented specie is larger than other represented species, you should repeat the simulation with this specie included in the represented species list. In turn, the mass fraction of all unrepresented elements should decrease.

Note

Note that **Dimension Reduction** is only available with ISAT. **Dimension Reduction** is initially comparable in speed to a simulation with the full mechanism, but iterations become significantly faster at later times when the ISAT table is populated.

For information about the theory of this option, see Chemical Mechanism Dimension Reduction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 22: Modeling Engine Ignition

This chapter discusses how to use the engine ignition models available in ANSYS FLUENT in the following sections. For information about the theory behind these ignition models, see "Engine Ignition" in the Theory Guide.

22.1. Spark Model
22.2. Autoignition Models
22.3. Crevice Model

22.1. Spark Model

The spark model in ANSYS FLUENT will be described in the context of the premixed turbulent combustion model. For information regarding the theory of this model, see Spark Model in the Theory Guide. Information regarding the use of this model is detailed in the following section:

22.1.1. Using the Spark Model
22.1.2. Using the ECFM Spark Model

22.1.1. Using the Spark Model

You can model a single spark or multiple sparks. To activate the spark model, perform the following steps:

1. Select **Transient** from the **Time** list in the **General** task page.
2. Select an appropriate reaction model in the **Species Model** dialog box.

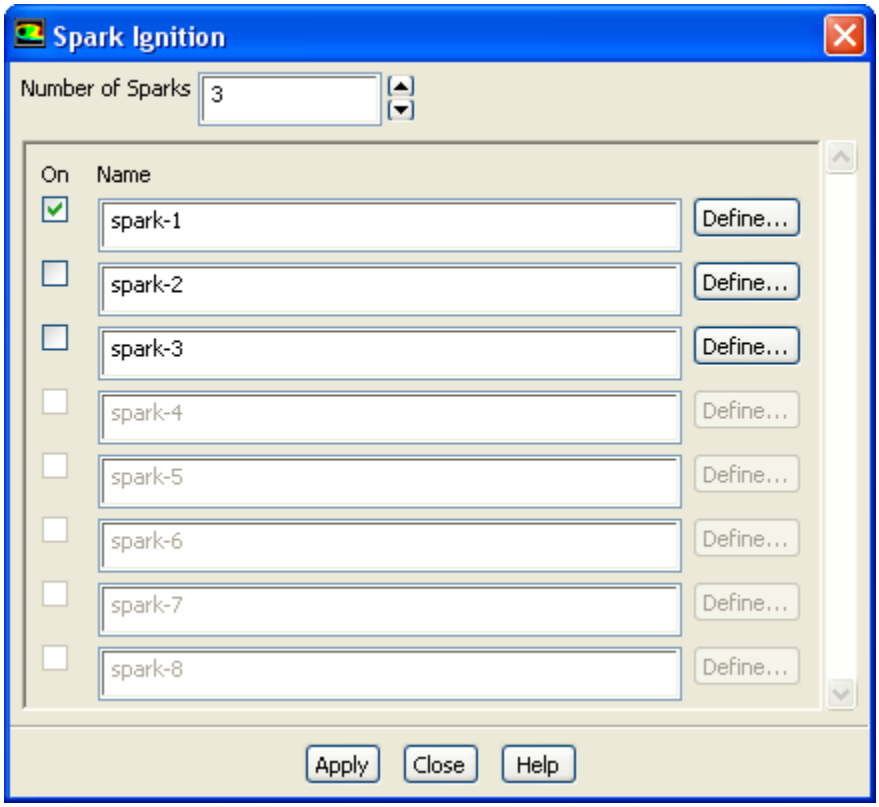
   Models → **Species** → **Edit...**

3. Select **Species Transport** under **Model** in the **Species Model** dialog box and enable **Volumetric** under **Reactions**. Or you can select the **Premixed Model** with the **C Equation** or **G Equation** model enabled. See *Setting Up the C-Equation and G-Equation Models* (p. 896) for more information.
4. The **Spark Ignition** model will now appear in the **Models** task page. Select **Spark Ignition** and click **Edit...**. This will open the **Spark Ignition** dialog box , as shown in *Figure 22.1* (p. 930).

   Models → **Spark Ignition** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

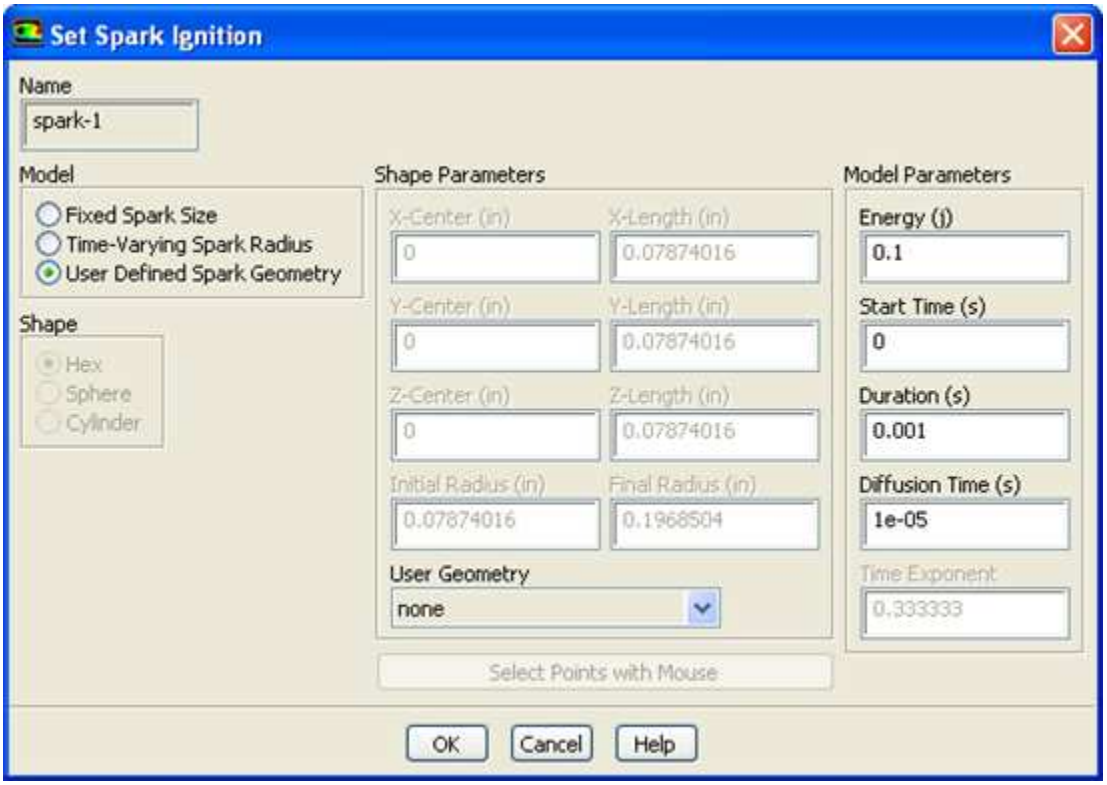
Figure 22.1 The Spark Ignition Dialog Box



5. Specify the **Number of Sparks** you would like to include in your simulation. You can define up to 16 sparks.
6. While you can define several sparks, you can choose which ones to activate using the **On** option.
7. Enter the **Name** of the spark, or simply keep the default name.
8. Click the **Define...** button to open the **Set Spark Ignition** dialog box (*Figure 22.2* (p. 931)), where you will set the parameters of the selected spark ignition models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.2 The Set Spark Ignition Dialog Box



9. Define the spark model as **Fixed Spark Size**, **Time-Varying Spark Radius**, or **User Defined Spark Geometry**.

When the **Fixed Spark Size** is enabled (*Figure 22.2* (p. 931)), you can select the shape of the spark to be spherical, cylindrical, or hexahedral in three dimensional simulations, or circular or quadrilateral in two dimensional simulations. Depending on the shape selected, appropriate inputs are highlighted or grayed out.

- To define a spherical spark, the shape parameters can be selected by clicking the **Select Points with Mouse** button at the bottom of the dialog box, and highlighting the center and radius of the spark kernel.
- Enter **Energy**, **Start Time**, **Duration**, and **Diffusion Time** in the **Set Spark Ignition** dialog box.

Important

When the in-cylinder model is turned on, the **Start Time** is entered in crank angle degrees instead of seconds, while the spark **Duration** is still in seconds.

- While the **Energy** input is in Joules by default, you can redefine the units as needed. The rate of energy input into the domain is constant so that the total energy will be evenly distributed over the duration that you set. The **Energy** input in the spark model should result in an appropriate temperature rise in the cell that is high enough to initiate combustion. The **Energy** input is only a model parameter and does not reflect energy input in actual automotive ignition systems, which typically range between 50 and 150 millijoules.

If you select the **Time-Varying Spark Radius** model, you will need to specify the **Energy**, **Start Time**, **Duration**, **Diffusion Time**, and **Time Exponent**. The spark is assumed spherical (in 3D and circular in 2D) and will grow from an **Initial Radius**, r_0, to a **Final Radius**, r_f, over the spark **Duration**, with a cube root depend-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

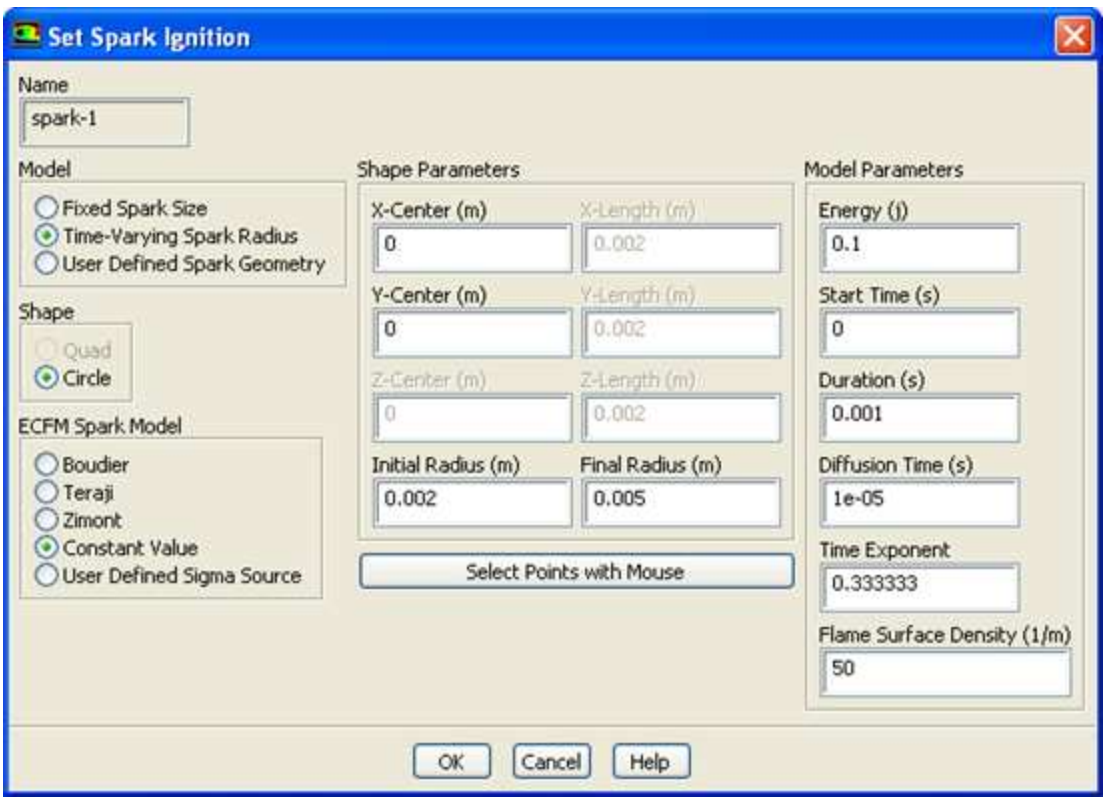
ence on time so that the radius will grow faster at the beginning and more slowly near the end. This time-dependent behavior is consistent with experimental findings [32] (p. 2210). The **Time-Varying Spark Radius** option is recommended as it has been found to be less sensitive to model parameters.

If you select **User Defined Spark Geometry** model, you can hook a user-defined function to define custom spark kernel volume shapes, which becomes selectable from the **User Geometry** drop-down list. For details about this user-defined function, please refer to DEFINE_SPARK_GEOM in the UDF Manual. In addition, you will need to specify the model parameters, such as **Energy**, **Start Time**, **Duration**, and **Diffusion Time**.

22.1.2. Using the ECFM Spark Model

When the **Extended Coherent Flamelet Model** (see *Setting Up the Extended Coherent Flame Model* (p. 899)) is selected in the **Species Model** dialog box, the **Set Spark Ignition** dialog box will exhibit additional **ECFM Spark Model** options, as shown in *Figure 22.3* (p. 932). These options are described in detail in ECFM Spark Model Variants in the Theory Guide.

Figure 22.3 The Set Spark Ignition Dialog Box Displaying the ECFM Spark Model Options



In addition to specifying the variables mentioned in *Using the Spark Model* (p. 929), you can

- Select the **Fixed Spark Size** model and select one of the three available **ECFM Spark Model** options:
  - **Zimont**: Specify the **Energy**, **Start Time**, **Duration**, and **Diffusion Time**.
  - **Constant Value**: Specify the **Energy**, **Start Time**, **Duration**, **Diffusion Time**, and the **Flame Surface Density**.
  - **User Defined Sigma Source**: Select the hooked UDF from the **User Sigma Source** drop-down list to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel. For more detail about this user-defined function, please refer to Hooking DEFINE_ECFM_SPARK_SOURCE UDFs in the UDF Manual.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **Time-Varying Spark Radius** model and select one of the five available **ECFM Spark Model** options:
  - **Boudier**: Specify the **Energy**, **Start Time**, **Duration**, **Diffusion Time**, and **Time Exponent**.
  - **Teraji**: Specify the **Energy**, **Start Time**, **Duration**, and **Diffusion Time**.
  - **Zimont**: Specify the **Energy**, **Start Time**, **Duration**, **Diffusion Time**, and **Time Exponent**.
  - **Constant Value**: Specify the **Energy**, **Start Time**, **Duration**, **Diffusion Time**, **Time Exponent**, and the **Flame Surface Density**.
  - **User Defined Sigma Source**: Select the hooked UDF from the **User Sigma Source** drop-down list to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel. For more detail about this user-defined function, please refer to Hooking DEFINE_ECFM_SPARK_SOURCE UDFs in the UDF Manual.
- Select the **User Defined Spark Geometry** model and select one of the two available **ECFM Spark Model** options:
  - **Constant Value**: Specify the **Energy**, **Start Time**, **Duration**, **Diffusion Time**, and the **Flame Surface Density**.
  - **User Defined Sigma Source**: Select the hooked UDF from the **User Sigma Source** drop-down list to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel. For more detail about this user-defined function, please refer to Hooking DEFINE_ECFM_SPARK_SOURCE UDFs in the UDF Manual.

22.2. Autoignition Models

Autoignition phenomena in engines are due to the effects of chemical kinetics of the reacting flow inside the cylinder. There are two types of autoignition models considered in ANSYS FLUENT:

- knock model in spark-ignited (SI) engines
- ignition delay model in diesel engines

For information regarding the theory behind autoignition models, see Autoignition Models in the Theory Guide. *Using the Autoignition Models* (p. 933) describes how to use the autoignition models in ANSYS FLUENT.

22.2.1. Using the Autoignition Models

To activate the autoignition model, perform the following steps:

1. Select **Transient** from the **Time** list in the **General** task page.
2. Select an appropriate reaction model in the **Species Model** dialog box.

   Models → **Species** → **Edit...**
3. The models in the **Species Model** dialog box that are compatible with the autoignition model are **Species Transport**, **Premixed Combustion**, and **Partially Premixed Combustion**.

Important

If you select **Species Transport**, you must also enable the **Volumetric** option in the **Reactions** group box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

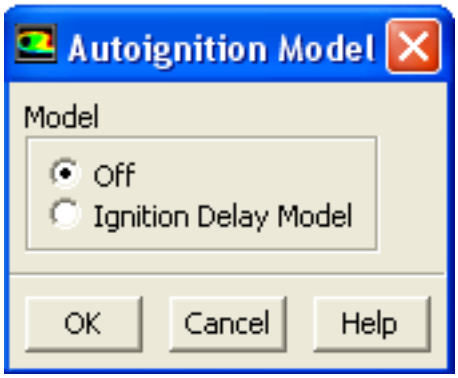
The **Premixed Combustion** and **Partially Premixed Combustion** models are only available for turbulent flows using the pressure-based solver.

4. The **Autoignition** model will now appear in the **Models** task page.

Models → **Autoignition** → **Edit...**

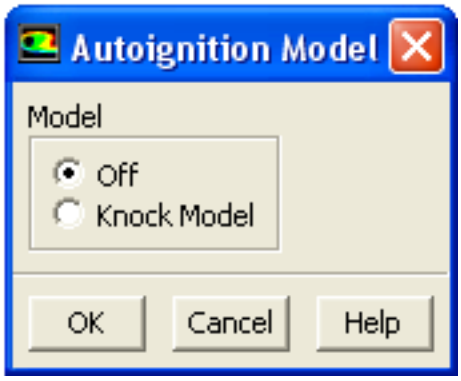
- If **Species Transport** is selected in the **Species Model** dialog box, you can only select the **Ignition Delay Model**.

Figure 22.4 The Ignition Delay Model in the Autoignition Model Dialog Box



- If **Premixed Combustion** is selected in the **Species Model** dialog box, you can only select the **Knock Model**.

Figure 22.5 The Knock Model in the Autoignition Model Dialog Box

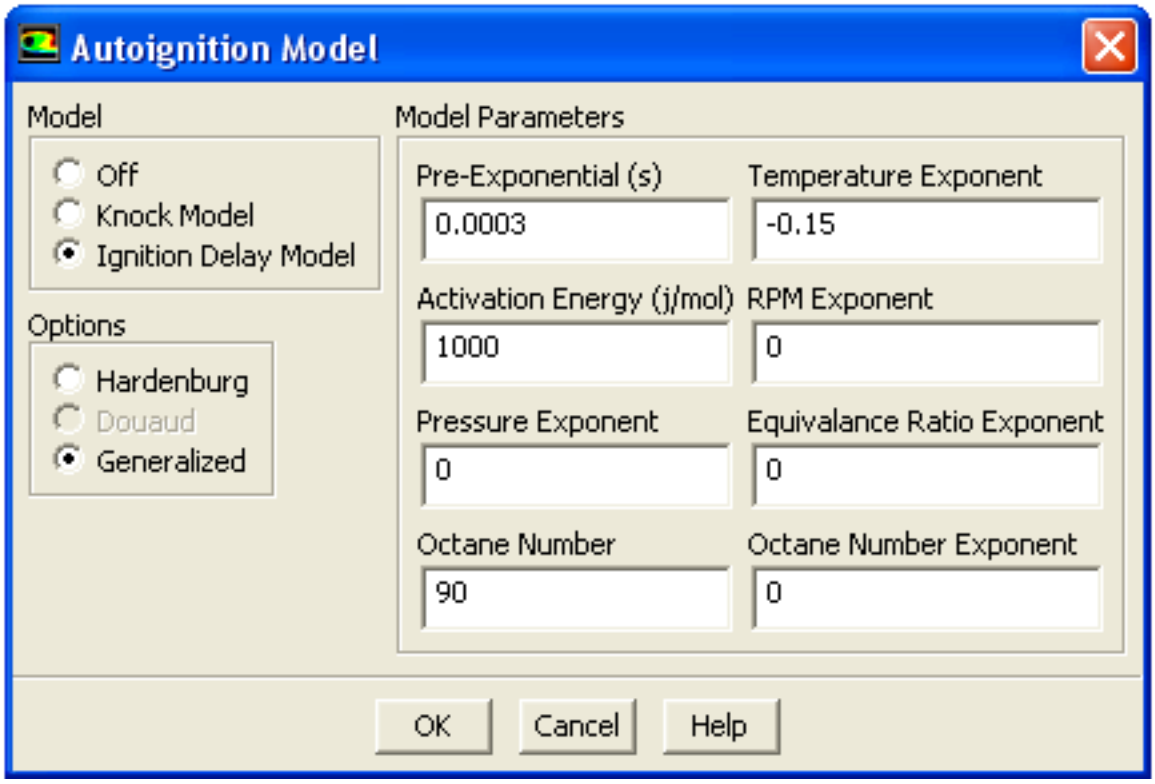


- If **Partially Premixed Combustion** is selected in the **Species Model** dialog box, you can select either the **Knock Model** or the **Ignition Delay Model**.

5. When the **Ignition Delay Model** is enabled, the dialog box expands to include the modeling parameters for this model (*Figure 22.6* (p. 935)). The two correlation options that exist with this model are **Hardenburg** and **Generalized**. Depending on which correlation option is selected, the appropriate modeling parameters will appear in the dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.6 The Ignition Delay Model for the Partially Premixed Combustion Model

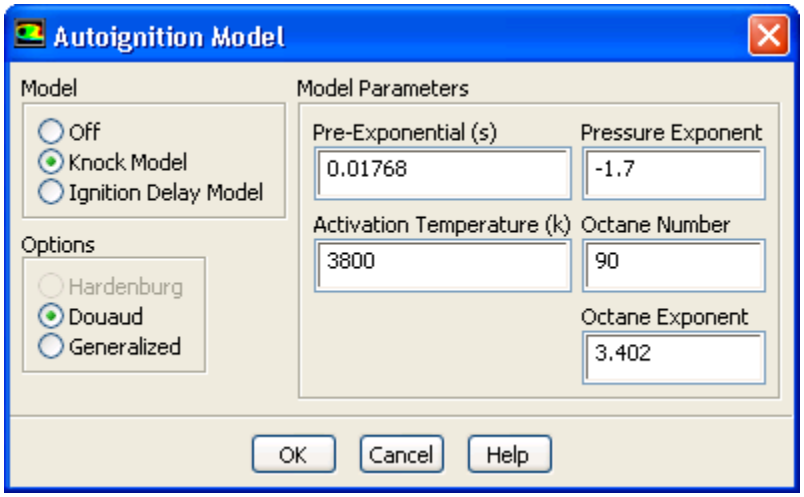


- The **Hardenburg** option is typically used for heavy duty diesel engines. A **Fuel Species** is selected from the drop-down list and the **Pre-Exponential**, **Pressure Exponent**, **Activation Energy**, and **Cetane Number** are entered using the GUI. Default values of these parameters can be found in Table 13.1: "Default Values of the Variables in the Hardenburg Correlation" in the Theory Guide.
- The **Generalized** option is described by Equation 13–13 in the Theory Guide. Similarly to the **Hardenburg** option, a **Fuel Species** is selected from the drop-down list and the **Pre-Exponential**, **Temperature Exponent**, **Activation Energy**, **RPM Exponent**, **Pressure Exponent**, **Equivalence Ratio Exponent**, **Octane Number**, and **Octane Number Exponent** are entered using the GUI.

6. When the **Knock Model** is enabled, the dialog box expands to include modeling parameters for this model (*Figure 22.7* (p. 936)). The two correlation options that exist with this model are **Douaud** and **Generalized**. Depending on which correlation option is selected, the appropriate modeling parameters will appear in the dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.7 The Knock Model with the Partially Premixed Combustion Model Enabled



- The **Douaud** option is used for knock in SI engines. The modeling parameters that are specified in the GUI for this option are the **Pre-Exponential**, **Pressure Exponent**, **Activation Temperature**, **Octane Number**, and **Octane Exponent** (Equation 13–12 in the Theory Guide).
- The **Generalized** option (Equation 13–13 in the Theory Guide) in the knock model requires the same parameters as in the ignition delay model.

22.3. Crevice Model

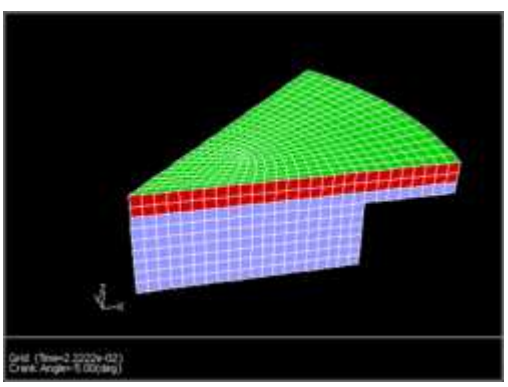
For information regarding the theory behind the crevice model, see Crevice Model in the Theory Guide. Using the crevice models in ANSYS FLUENT are described in the following sections:

22.3.1. Using the Crevice Model
22.3.2. Crevice Model Solution Details
22.3.3. Postprocessing for the Crevice Model

22.3.1. Using the Crevice Model

An optical experimental engine [*19*] (p. 2210) is used below to show a working example of how to use the crevice model as it is implemented in ANSYS FLUENT. The mesh at ten crank angle degrees before top center is shown in *Figure 22.8* (p. 936).

Figure 22.8 Experimental Engine Mesh



The following example shows the necessary steps to enable the crevice model for a typical in-cylinder flow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the > prompt, enter the `define/models` menu by using the following text command:

   `define → models`

2. Enable the crevice model, as follows:

```
/define/models    crevice-model?

Enable crevice model? [no] yes
/define/models
acoustics/                  multiphase/                      species/
addon-module                noniterative-time-advance?     steady?
axisymmetric?               nox?                               unsteady-1st-order?
crevice-model-controls/    radiation/                        unsteady-2nd-order?
crevice-model?              solidification-melting?        viscous/
dpm/                          solver/
energy?                       soot?
frozen-flux?               sox?
```

3. Enter the ring pack geometry:

```
/define/models    crevice-model-controls

Cylinder bore (m) [0.1] 0.1397
Piston to bore clearance (m) [3.0e-5] 5.08e-05
Piston crevice temperature (K) [400] 433
Piston sector angle (deg) [360] 45
Ring discharge coefficient [0.8] 0.7
Pressure in crankcase (exit pressure) (Pa) [101325]
Write out crevice data to a file? [no] yes

output file name ["crev.out"]

 Available wall threads are: (wall.1 wall wall-8)
 Leaking wall []   wall.1

Shared boundary [] wall-8
   Selected boundary threads : (wall.1 wall-8)
  Use these zones? [yes]   yes

Solve crevice model ? [no] yes
Number of rings [3]
            Width of ring number 0 is: [0.00375]
        Thickness of ring number 0 is: [0.0015]
          Spacing of ring number 0 is: [0.008]
   Land Length for ring number 0 is: [0.00391]
         Top Gap of ring number 0 is: [6e-05]
      Middle Gap of ring number 0 is: [4e-05]
      Bottom Gap of ring number 0 is: [6e-05]

            Width of ring number 1 is: [0.00375]
        Thickness of ring number 1 is: [0.0015]
          Spacing of ring number 1 is: [0.008]
   Land Length for ring number 1 is: [0.00391]
         Top Gap of ring number 1 is: [6e-05]
      Middle Gap of ring number 1 is: [4e-05]
      Bottom Gap of ring number 1 is: [6e-05]

            Width of ring number 2 is: [0.00375]
        Thickness of ring number 2 is: [0.0015]
          Spacing of ring number 2 is: [0.00391]
   Land Length for ring number 2 is: [0.00391]
         Top Gap of ring number 2 is: [6e-05]
      Middle Gap of ring number 2 is: [4e-05]
      Bottom Gap of ring number 2 is: [6e-05]


Initial conditions in ring pack
Pressure 1 is: [4600623.5]
Pressure 2 is: [4173522.5]
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Pressure  3  is:  [3689110.5]
Pressure  4  is:  [3130620]
Pressure  5  is:  [2214841.8]
```

A fast way to set up multiple rings in the ring pack is to specify only one ring and enter the geometry. Once the ring geometry is entered, invoke the `crevice-model- controls` menu a second time and specify the number of rings desired. When the number of rings changes, the geometry from the first ring is copied to all subsequent rings. Default values can be taken for the rest of the way through the menu structure.

A summary of the crevice model is printed out by entering the `(crevice-summary)` command at the command prompt:

```
>(crevice-summary)
                        crevice/n-rings  :  3
                     crevice/ring-width  :  (0.00375  0.00375  0.00375)
                crevice/ring-thickness  :  (0.0015  0.0015  0.0015)
                      crevice/ring-mass  :  (0.00375  0.00375  0.00375)
                  crevice/ring-spacing  :  (0.008  0.008  0.00391)
                   crevice/land-length  :  (0.00391  0.00391  0.00391)
                  crevice/top-ring-gap  :  (6e-05  6e-05  6e-05)
                  crevice/mid-ring-gap  :  (4e-05  4e-05  4e-05)
                  crevice/bot-ring-gap  :  (6e-05  6e-05  6e-05)
            crevice/piston-temperature  :  433
                  crevice/sector-angle  :  45
                    crevice/mid-gap-cd  :  0.7
                 crevice/exit-pressure  :  101325
                        crevice/threads  :  (5  6)
            names  of  crevice/threads  :  (wall.1  wall-8)
                crevice/unit-roundoff  :  5.9604645e-08
         crevice/piston-bore-clearance  :  5.08e-05
                         crevice/write?  :  #t
                   crevice/output-file  :  crev.out
                         crevice/solve?  :  #t
                      crevice/enabled?  :  #t
                     crevice/pressures  :  (4600623.5  4173522.5  3689110.5  3130620  2214841)
```

22.3.2. Crevice Model Solution Details

The under-relaxation factor for the crevice model source terms can be found in the **Solution Controls** task page. The default value for **Crevice Model Sources** is 0.8, which has been found to work well for motored engine simulations. Once the crevice model is enabled, the solution proceeds normally.

Solution Controls

Solution Initialization

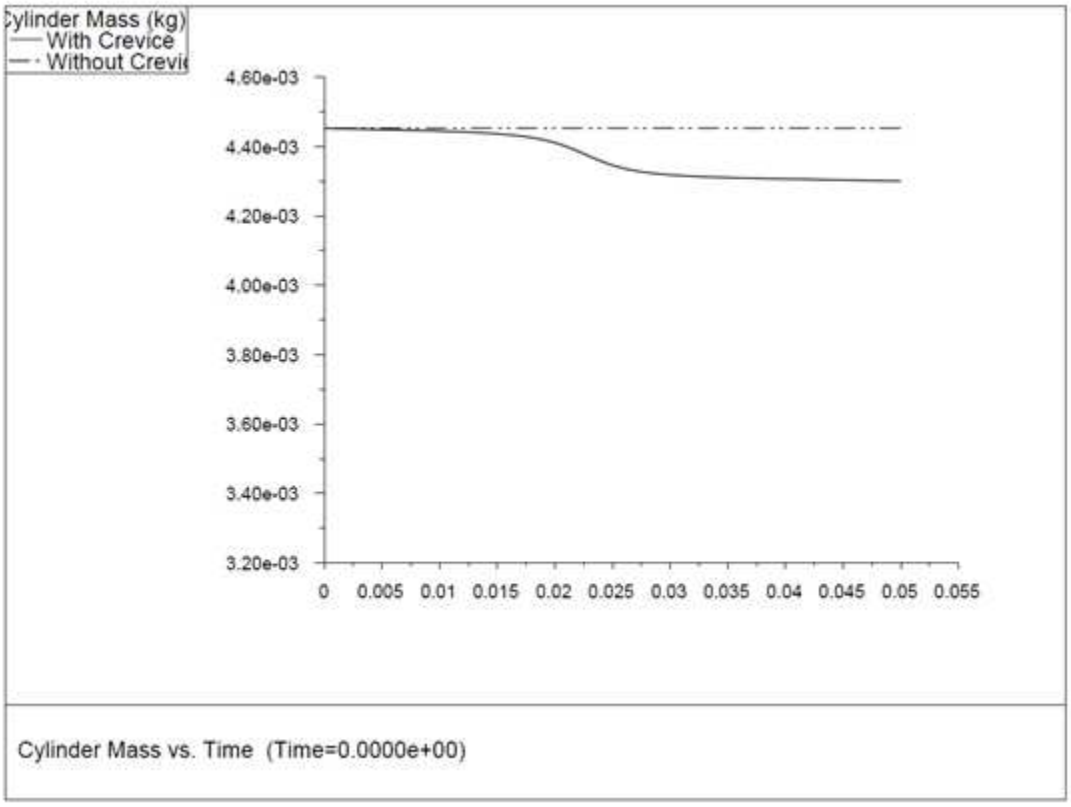
Run Calculation

22.3.3. Postprocessing for the Crevice Model

A plot of cylinder mass with and without the crevice model during the motored engine simulation is shown in *Figure 22.9* (p. 939). The rate of mass loss from the crevice is proportional to the pressure difference between the cylinder and the crankcase pressure defined in the text interface.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

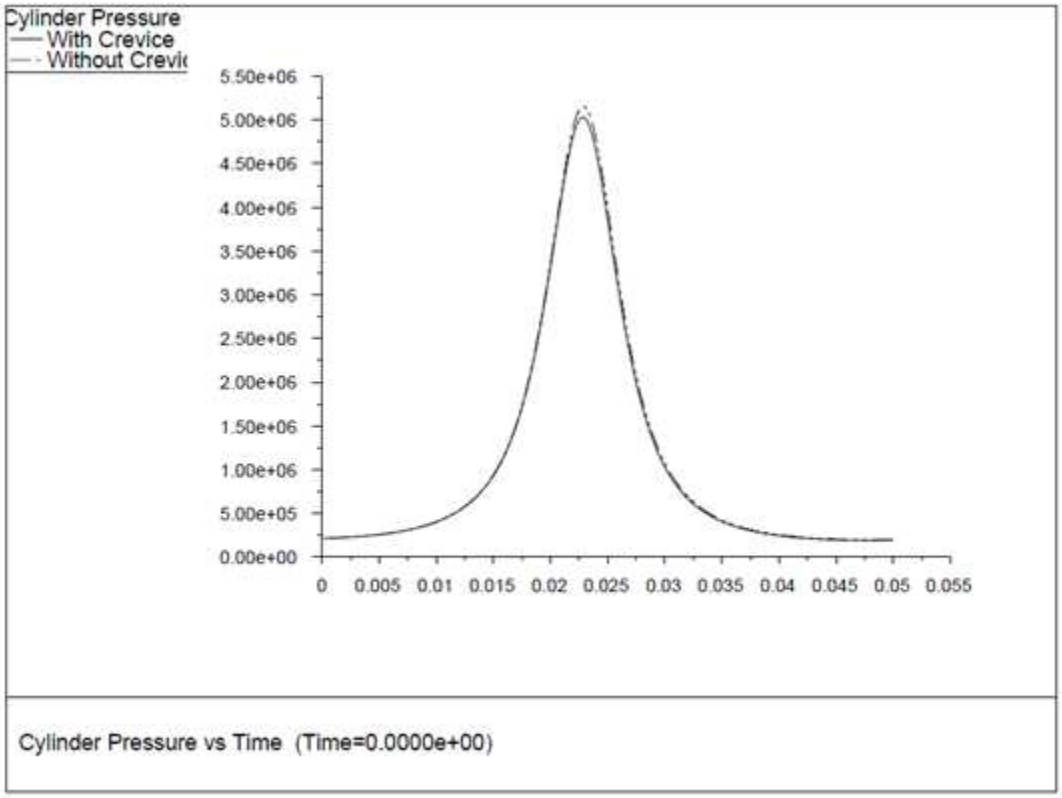
Figure 22.9 Cylinder Mass vs. Crank Angle



A plot of cylinder pressure with and without the crevice model for the same engine simulation is shown in *Figure 22.10* (p. 940). The effect of the mass loss from the crevice is to lower the peak pressure in proportion to the total mass loss from the cylinder.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.10 Cylinder Pressure vs. Crank Angle



22.3.3.1. Using the Crevice Output File

The pressure in the top ring land is defined as the cylinder pressure (i.e., the pressure in the cells defining the ring landing). Intermediate pressures are available at any point during the ANSYS FLUENT session through the `(crevice-summary)` command as previously shown. If the optional data file output is chosen in the `crevice-model-controls`, the intermediate pressures in the defined crevices are printed to the file `crev.out` at the start of each new time step. The format of the file is as follows:

```
 # crank (deg) data-press[0...1...2...3...4...5...6] total_mdot
1.95500e+02 2.16650e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 0.0
1.96000e+02 2.09945e+05 1.06794e+05 1.81553e+05 1.04111e+05 1.48582e+05 1.02202e+05 1.01325e+05 -1.6
1.96500e+02 2.17787e+05 1.13070e+05 1.88242e+05 1.07960e+05 1.53544e+05 1.03526e+05 1.01325e+05 -1.6
1.97000e+02 2.17434e+05 1.19065e+05 1.88060e+05 1.11705e+05 1.53475e+05 1.04830e+05 1.01325e+05 -1.6
1.97500e+02 2.17652e+05 1.24777e+05 1.88299e+05 1.15286e+05 1.53668e+05 1.06081e+05 1.01325e+05 -1.6
1.98000e+02 2.17937e+05 1.30215e+05 1.88594e+05 1.18711e+05 1.53900e+05 1.07283e+05 1.01325e+05 -1.6
```

where the first column is the current flow time (or crank angle), and the next $n_{cv}+2$ columns are the ring pressures (where n_{cv} is the number of crevice volumes, or $2n_r-1$), including the face pressure on the crevice cell, and the defined pressure at the crevice exit. The final column is the mass flow past the top ring. This file is currently formatted so that it can be read into the free **Gnuplot** plotting package, which is available at `www.gnuplot.info`.

To read the crevice output file into ANSYS FLUENT as a data file, you will need to put each column of the crevice output file in its own individual file. The first three lines of each column of the data file should be of the following form:

```
"Title"
"X-Label" "Y-Label"
0 0 0 0
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where the title, x-label, and y-label strings are enclosed by double quotes and the third line of the file contains four zeros. The lines following the first three lines of the file are the columns you wish to plot. For example, to plot column 1 versus column 3 of the crevice model output file in ANSYS FLUENT, you would enter the following commands in a UNIX terminal:

```
cat > crev_col_1_3.dat
"Column 1 vs Column 3"
"Crank Angle (deg)" "Pressure behind ring 1 (Pa)"
0 0 0 0
ctrl-d
```

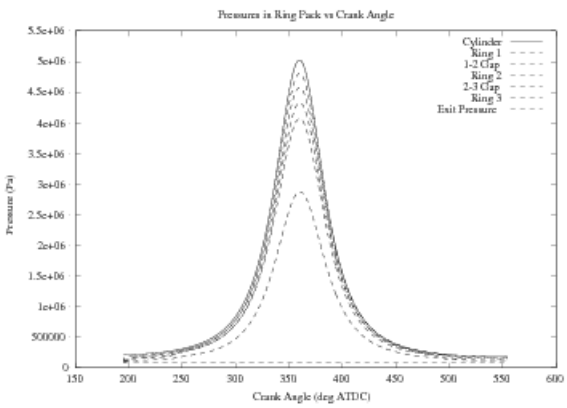
where `ctrl-d` is the end-of-file character made holding down the `<Ctrl>` key and pressing `d`. To append columns 1 and 3 to this file, enter the following:

```
tail +2 crev.out | awk '{print $1, $3}' >> crev_col_1_3.dat
```

The file `crev_col_1_3.dat` can now be read into ANSYS FLUENT using the *File XY Plot Dialog Box* (p. 2016). See *XY Plots of File Data* (p. 1472) for details about creating x- y plots. For Windows users, the file `crev.out` can be imported into Excel for plotting purposes without any modification.

A **Gnuplot** plot of the pressure in the ring pack crevices for the above engine simulation is shown in *Figure 22.11* (p. 941). After an initial transient period where the flows in the network settle down, *Figure 22.11* (p. 941) shows that the pressure in the ring crevices follows the cylinder pressure in form, though with pressure magnitudes that are controlled by the ring pack geometry.

Figure 22.11 Crevice Pressures



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 23: Modeling Pollutant Formation

This chapter discusses how to use the models available in ANSYS FLUENT for modeling pollutant formation. For information about the theory behind the models in ANSYS FLUENT, see "Pollutant Formation" in the Theory Guide.

Information is presented in the following sections:

23.1. NOx Formation
23.2. SOx Formation
23.3. Soot Formation
23.4. Using the Decoupled Detailed Chemistry Model

23.1. NOx Formation

The following sections describe how to use the NO_x models in ANSYS FLUENT. For information about the theory behind the NO_x models in ANSYS FLUENT, see NOx Formation in the Theory Guide.

23.1.1. Using the NOx Model
23.1.2. Solution Strategies
23.1.3. Postprocessing

23.1.1. Using the NOx Model

23.1.1.1. Decoupled Analysis: Overview

NO_x concentrations generated in combustion systems are generally low. As a result, NO_x chemistry has negligible influence on the predicted flow field, temperature, and major combustion product concentrations. It follows that the most efficient way to use the NO_x model is as a postprocessor to the main combustion calculation.

The recommended procedure is as follows:

1. Calculate your combustion problem using ANSYS FLUENT as usual.

Important

The premixed combustion model is not compatible with the NO_x model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you plan to use the ANSYS FLUENT SNCR model for NO_x reduction, you will first need to include ammonia or urea (depending upon which reagent is employed) as a fluid species in the main combustion calculation and define appropriate ammonia injections, as described later in this section. See *Defining the Species in the Mixture* (p. 806) for details about adding species to your model and *Setting Initial Conditions for the Discrete Phase* (p. 1022) for details about creating injections.

2. Enable the desired NO_x models (thermal, prompt, fuel, and/or N_2O intermediate NO_x, with or without reburn), define the fuel streams (for prompt NO_x and fuel NO_x only), and set the appropriate parameters, as described in this section.

   Models → **NOx** → **Edit...**

3. Define the boundary conditions for NO (and HCN, NH_3, or N_2O, if necessary) at flow inlets.

   Boundary Conditions

4. In the *Equations Dialog Box* (p. 1904), turn off the solution of all variables except species NO (and HCN, NH_3, or N_2O, based on the model selected).

   Solution Controls → **Equations...**

5. Perform calculations until convergence (i.e., until the NO—and HCN, NH_3, or N_2O, if solved—species residuals are below 10^{-6}) to ensure that the NO and HCN or NH_3 concentration fields are no longer evolving.

   Run Calculation

6. Review the mass fractions of NO (and HCN, NH_3, or N_2O) by generating graphical plots or alphanumeric reports in the usual way.
7. Save a new set of case and data files, if desired.

   File → **Write** → **Case & Data...**

Inputs specific to the calculation of NO_x formation are explained in the remainder of this section.

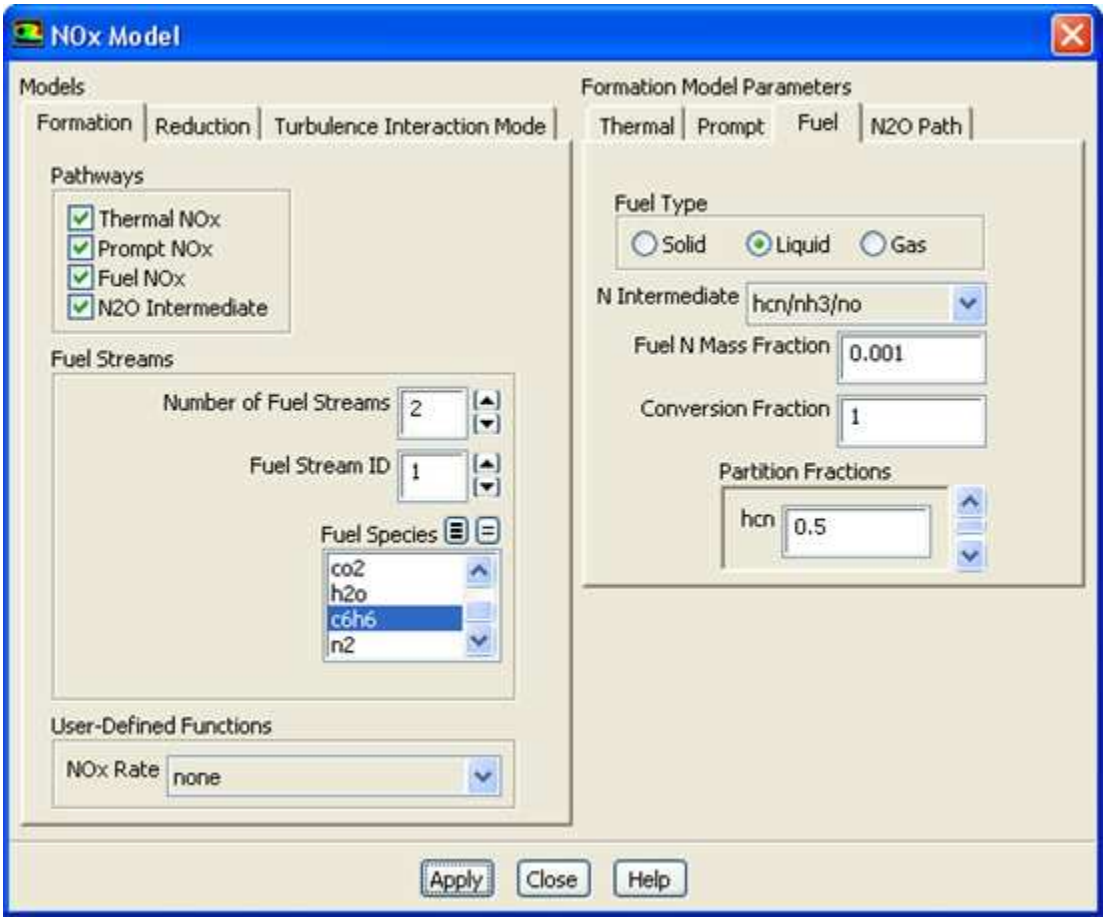
23.1.1.2. Enabling the NOx Models

To enable the NO_x models and set related parameters, you will use the *NOx Model Dialog Box* (p. 1706) (e.g., *Figure 23.1* (p. 945)).

Models → **NOx** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.1 The NOx Model Dialog Box



In the **Formation** tab, select the NO_x models under **Pathways** to be used in the calculation of the NO and HCN, NH_3, or N_2O concentrations:

- To enable thermal NO_x, turn on the **Thermal NOx** option.
- To enable prompt NO_x, turn on the **Prompt NOx** option.
- To enable fuel NO_x, turn on the **Fuel NOx** option.

Important

When using the non-premixed combustion model, the **Fuel NOx** option is only available if the DPM model is also enabled.

- To enable the formation of NO_x through an N_2O intermediate, turn on the **N2O Intermediate** option. (Note that the **N2O Intermediate** option will not appear until you have activated one of the other NO models listed above.)

Your selection(s) under **Pathways** will activate the calculation of thermal, prompt, fuel, and/or N_2O-intermediate NO_x in accordance with the chemical kinetic models described in Thermal NOx Formation through NOx Formation from Intermediate N2O in the Theory Guide. Mean NO formation rates will be computed directly from mean concentrations and temperature in the flow field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.1.1.3. Defining the Fuel Streams

ANSYS FLUENT allows you to define multiple fuel streams when you are modeling prompt or fuel NO_x formation. If either **Prompt NOx** or **Fuel NOx** is enabled in the **Pathways** group box in the **Formation** tab, perform the following steps:

1. Specify the **Number of Fuel Streams** in the **Fuel Streams** group box. You are allowed up to three separate fuel streams.
2. Define the first fuel stream.
   a. Select the fuel stream to be defined by using the arrow keys of the **Fuel Stream ID** text box.
   b. When modeling fuel NO_x formation in conjunction with the non-premixed combustion model (which requires that the discrete phase model be enabled as well), make a selection from the **PDF Stream** drop-down list (*Figure 23.2* (p. 947)) to define the species for this stream. You can select either the **primary** or **secondary** fuel stream species, as defined in the PDF table.

      Important

      Note that the **PDF Stream** drop-down list defines the species for the fuel NO_x calculations only.

   c. When modeling prompt NO_x formation or when using a combustion model other than non-premixed combustion, select the fuel species from the **Fuel Species** list. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10.

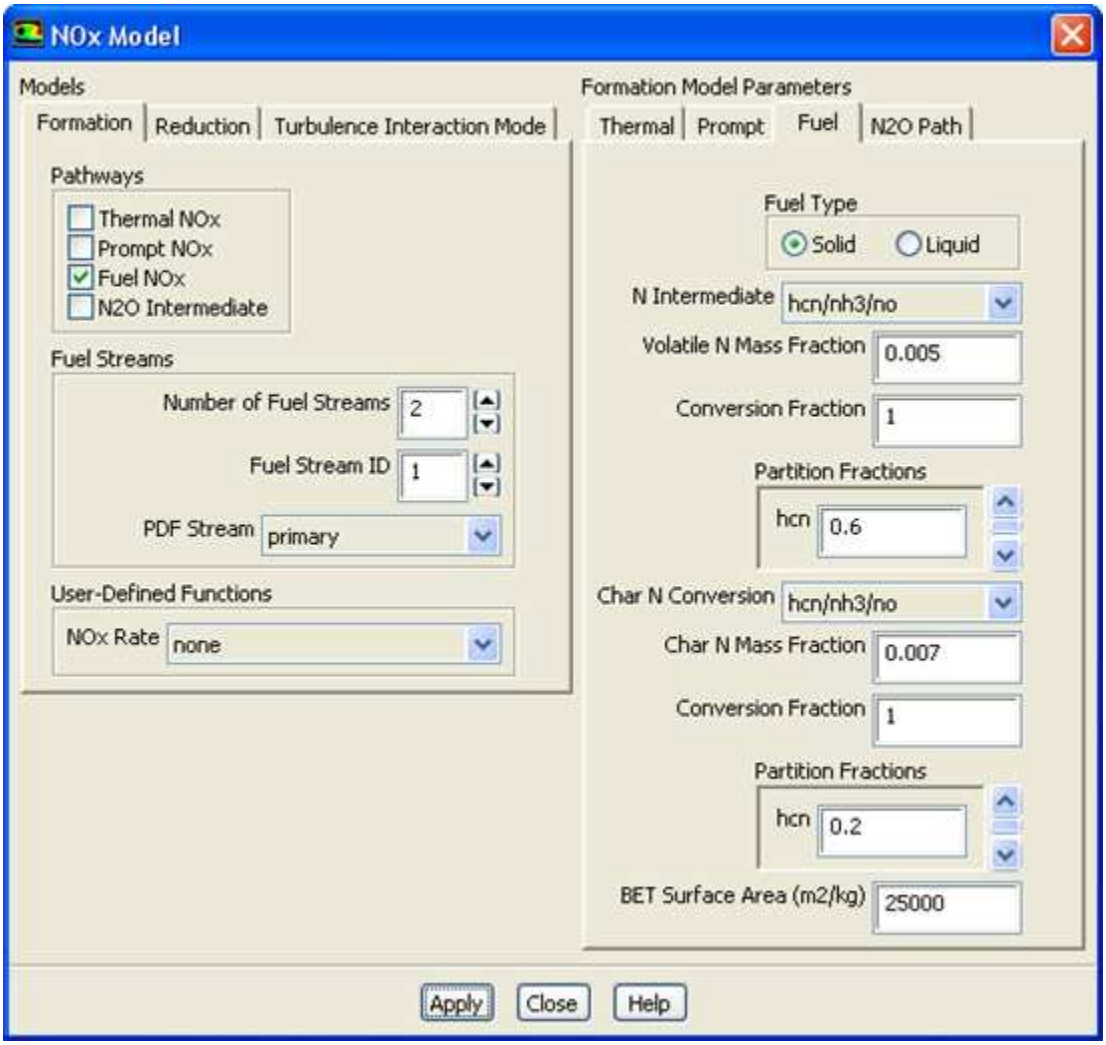
      Important

      Note that the **Fuel Species** selections define the species for the prompt NO_x calculations only when modeling non-premixed combustion.

   d. Set the other parameters associated with your selected pathway(s) in the **Prompt** and/or **Fuel** tabs under **Formation Model Parameters**. See *Setting Prompt NOx Parameters* (p. 949) and *Setting Fuel NOx Parameters* (p. 949) for details.
3. Repeat steps 2.(a)–2.(d) for each additional fuel stream.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.2 The NOx Model Dialog Box with Non-Premixed Combustion



Note that the following limitations apply when you are modeling fuel NO_x formation with multiple fuel streams, if more than one fuel stream has the same fuel type (as defined in the **Fuel Type** group box in the **Fuel** tab):

- You should only use multiple liquid fuel streams or multiple solid (coal) fuel streams when you have streams that have different destination species. Therefore, you must make sure that the injectors associated with the fuel streams specify these unique destination species correctly, as defined in the **Devolatilizing Species** drop-down list in the *Set Injection Properties Dialog Box* (p. 2097) (see *Defining Injection Properties* (p. 1040) for details).
- For multiple solid (coal) fuel streams, the fuel streams should have the same char-related parameter values in the **Fuel** tab — i.e., the **Char N Mass Fraction**, the **Partition Fractions** (for char N), and the **BET Surface Area** values. Note that even if different values are set for these char-related parameters, ANSYS FLUENT will only recognize those specified for the solid fuel stream with the lowest ID number, and then apply them to all of the other solid fuel streams.

For more information about the limitations associated with multiple fuel streams with the same fuel type, contact your ANSYS FLUENT support engineer.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that if you read a case file with NO_x settings that was set up in a version of ANSYS FLUENT previous to 12, you may need to make a selection for the fuel species. This step is only necessary when all of the following conditions are met:

- **Fuel NOx** is enabled in the **Pathways** group box.
- **Prompt NOx** is not enabled in the **Pathways** group box.
- **Solid** or **Liquid** is selected for **Fuel Type** in the **Fuel** tab.

Your fuel species selection should be made either in the **PDF Stream** drop-down list for non-premixed combustion, or in the **Fuel Species** list for all other combustion models.

23.1.1.4. Specifying a User-Defined Function for the NOx Rate

You can choose to specify a user-defined function for the rate of NO_x production. By default, the rate returned from the UDF is added to the rate returned from the standard NO_x production options, if any are selected. You also have the option of replacing any or all of ANSYS FLUENT's NO_x rate calculations with your own user-defined NO_x rate.

In addition to or instead of using the UDF to specify the NO_x rate, you can use it to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

To use a UDF to add a rate to ANSYS FLUENT's NO_x rate calculations, you must compile and load the desired function, and then select it from the **NOx Rate** drop-down list in the **User-Defined Functions** group box in the **Formation** tab. After you have selected the UDF, you have the following options:

- You can specify that your custom rate is added to the ANSYS FLUENT NO_x rate calculations, by retaining the default selection of **Add to FLUENT Rate** in the **UDF Rate** group box for the appropriate NO_x formation pathway(s) (e.g., in the **Fuel** tab).
- You can replace the ANSYS FLUENT NO_x rate calculations with your custom rate, by selecting **Replace FLUENT Rate** in the **UDF Rate** group box for the appropriate NO_x formation pathway(s) (e.g., in the **Fuel** tab).
- You can specify custom values for T_{max}, by selecting **user-defined** from the **Tmax Option** drop-down list in the **Turbulence Interaction Mode** tab.

See the UDF Manual for details about user-defined functions.

23.1.1.5. Setting Thermal NOx Parameters

The NO_x routines employ three methods for calculation of thermal NO_x (as described in Method 1: Equilibrium Approach in the Theory Guide). You will specify the method to be used in the **Thermal** tab, under **Formation Model Parameters** in the *NOx Model Dialog Box* (p. 1706):

- To choose the equilibrium method, select **equilibrium** in the **[O] Model** drop-down list.
- To choose the partial equilibrium method, select **partial-equilibrium** in the **[O] Model** or **[OH] Model** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

Important

Note that the urea model uses the $[\mathrm{OH}]$ model.

If you hooked a UDF in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace FLUENT Rate** to replace ANSYS FLUENT's thermal NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to FLUENT Rate** to add the custom NO_x rate produced by your UDF to ANSYS FLUENT's thermal NO_x rate calculations.

23.1.1.6. Setting Prompt NOx Parameters

Prompt NO_x formation is predicted using Equation 14–26 and Equation 14–28 in the Theory Guide. For each fuel stream specified in the **Fuel Stream ID** text box in the **Formation** tab, set the parameters in the **Prompt** tab under **Formation Model Parameters** in the *NOx Model Dialog Box* (p. 1706) in the following manner:

- Set the **Fuel Carbon Number** to specify the number of carbon atoms per fuel molecule.
- Set the **Equivalence Ratio** as follows:

$$\text{Equivalence Ratio} = \frac{\text{actual fuel-to-air ratio}}{\text{stoichiometric fuel-to-air ratio}} \tag{23–1}$$

If you hooked a UDF in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace FLUENT Rate** to replace ANSYS FLUENT's prompt NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to FLUENT Rate** to add the custom NO_x rate produced by your UDF to ANSYS FLUENT's prompt NO_x rate calculations.

23.1.1.7. Setting Fuel NOx Parameters

When using the fuel NO_x model, you must set the parameters in the **Fuel** tab under **Formation Model Parameters** for each fuel stream specified in the **Fuel Stream ID** text box in the **Formation** tab.

If you hooked a UDF in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace FLUENT Rate** to replace ANSYS FLUENT's fuel NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to FLUENT Rate** to add the custom NO_x rate produced by your UDF to ANSYS FLUENT's fuel NO_x rate calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If there is no NO_x rate UDF or if you selected **Add to FLUENT Rate**, you must define fuel parameters. To begin, specify the fuel type in the following manner:

- For solid fuel NO_x, select **Solid** under **Fuel Type**.
- For liquid fuel NO_x, select **Liquid** under **Fuel Type**.
- For gaseous fuel NO_x, select **Gas** under **Fuel Type**.

Note that you can use only one of the fuel types for a given fuel stream. The **Gas** option is available only when the **Species Transport** model is enabled (see *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801)).

23.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters

If you have selected **Gas** or **Liquid** as the **Fuel Type**, you will also need to specify the following:

- Select the intermediate species (**hcn**, **nh3**, or **hcn/nh3/no**) in the **N Intermediate** drop-down list.
- Set the correct mass fraction of nitrogen in the fuel (kg nitrogen per kg fuel) in the **Fuel N Mass Fraction** field.
- Specify the overall fraction of the fuel N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field. The **Conversion Fraction** for the **N Intermediate** has a default value of 1. Thus, any remaining N will not contribute to NO_x formation. This is based on the assumption that the remaining volatile N will convert to gas phase nitrogen. However, this has very little effect on the overall mass fraction of gas phase nitrogen. Therefore, you do not have to solve for nitrogen species when solving pollutant transport equations.
- If you selected **hcn/nh3/no** as the intermediate, specify the fraction of the converted fuel N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of fuel N that will become NO will be calculated by the remainder.

  Note that setting a partition fraction of 0 for both HCN and NH_3 is equivalent to assuming that all fuel N is converted to the final product NO, whereas a partition fraction of 0 for HCN and 1 for NH_3 is the same as selecting **nh3** as the intermediate.

ANSYS FLUENT will use Equation 14–31 and Equation 14–32 in the Theory Guide (for HCN) or Equation 14–42 and Equation 14–43 in the Theory Guide (for NH_3) to predict NO formation for a gaseous or liquid fuel.

Important

Note that there is a limitation that must be considered when defining more than one liquid fuel stream. See *Defining the Fuel Streams* (p. 946) for details.

23.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters

For solid (coal) fuel, ANSYS FLUENT will use Equation 14–55 and Equation 14–56 in the Theory Guide (for HCN) or Equation 14–62 and Equation 14–63 in the Theory Guide (for NH_3) to predict NO formation. Several inputs are required for the coal fuel NO_x model as follows:

- Select the intermediate species (**hcn**, **nh3**, or **hcn/nh3/no**) in the **N Intermediate** drop-down list.
- Specify the mass fraction of nitrogen in the volatiles in the **Volatile N Mass Fraction** field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the overall fraction of the volatile N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field.
- If you selected **hcn/nh3/no** as the volatile N intermediate, specify the fraction of the converted volatile N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of volatile N that will become NO will be calculated by the remainder.
- Select the char N conversion path from the **Char N Conversion** drop-down list as **no**, **hcn**, **nh3**, or **hcn/nh3/no**. Note that **hcn** or **nh3** can be selected only if the same species has been selected as the intermediate species in the **N Intermediate** drop-down list.
- Specify the mass fraction of nitrogen in the char in the **Char N Mass Fraction** field.
- Specify the overall fraction of the char N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field.
- If you selected **hcn/nh3/no** as the char N conversion, specify the fraction of the converted char N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of char N that will become NO will be calculated by the remainder.
- Define the BET internal pore surface area (see BET Surface Area in the Theory Guide for details) of the particles in the **BET Surface Area** field.

Important

Note that there are limitations that must be considered when defining more than one solid fuel stream. See *Defining the Fuel Streams* (p. 946) for details.

The following equations are used to determine the mass fraction of nitrogen in the volatiles and char:

$$\dot{m}_{N_{v/c}} = \dot{m}_{v/c} * mf_{N_{v/c}} \tag{23–2}$$

where

$\dot{m}_{N_{v/c}}$ = rate of release of fuel nitrogen in kg/s

$\dot{m}_{v/c}$ = rate of release of volatiles (v) or char (c) in kg/s

$mf_{N_{v/c}}$ = mass fraction of nitrogen in volatiles or char

Let

TN_{fuel} = total nitrogen mass fraction in daf coal (i.e., from daf ultimate analysis)

N_{split} = char nitrogen as a fraction of total nitrogen

F_{vol} = mass fraction of volatiles in daf coal

F_{char} = mass fraction of char in daf coal

Then the following should hold:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$F_{vol}+F_{char}=1 \tag{23–3}$$

$$\frac{F_{char}*mf_{N_c}}{TN_{fuel}}=N_{split} \tag{23–4}$$

$$F_{vol}*mf_{N_v}+F_{char}*mf_{N_c}=TN_{fuel} \tag{23–5}$$

$$mf_{N_v}=\left(1-N_{split}\right)*\frac{TN_{fuel}}{F_{vol}} \tag{23–6}$$

$$mf_{N_c}=N_{split}*\frac{TN_{fuel}}{F_{char}} \tag{23–7}$$

23.1.1.8. Setting N2O Pathway Parameters

The formation of NO through an $\mathrm{N_2O}$ intermediate can be predicted by two methods. You will specify the method to be used in the **N2O Path** tab.

- To choose the quasi-steady state method, select **quasi-steady** in the **N2O Model** drop-down list.

Important

The transport equation for the species $\mathrm{N_2O}$ will not be solved for $\mathrm{N_2O}$. However, $\mathrm{N_2O}$ will be updated at every iteration. Therefore, the residual values that appear for $\mathrm{N_2O}$ are always zero. Do not be alarmed if the solver keeps printing zero at each iteration.

- To choose the simplified form of the $\mathrm{N_2O}$-intermediate mechanism, select **transported-simple** in the **N2O Model** drop-down list. Here, the species $\mathrm{N_2O}$ is added to the list of pollutant species, and its mass fraction is solved via a transport equation.

The atomic O concentration will be calculated according to the thermal NO_x **[O] Model** that you have specified previously. If you have not selected the **Thermal NOx** pathway, then you will be given the option to specify an **[O] Model** for the $\mathrm{N_2O}$ pathway calculation. The same three options for the thermal NO_x **[O] Model** will be the available options.

If you hooked a UDF in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace FLUENT Rate** to replace the NO_x rate calculated by ANSYS FLUENT using $\mathrm{N_2O}$ intermediates with the custom NO_x rate produced by your UDF.
- Select **Add to FLUENT Rate** to add the custom NO_x rate produced by your UDF to the NO_x rate calculated by ANSYS FLUENT using $\mathrm{N_2O}$ intermediates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.1.1.9. Setting Parameters for NOx Reburn

To enable NO_x reduction by reburning, click the **Reduction** tab in the *NOx Model Dialog Box* (p. 1706) and enable the **Reburn** option under **Methods**. In the expanded portion of the dialog box, as shown in *Figure 23.3* (p. 953), click the **Reburn** tab under **Reduction Method Parameters**, where you can choose from the following options:

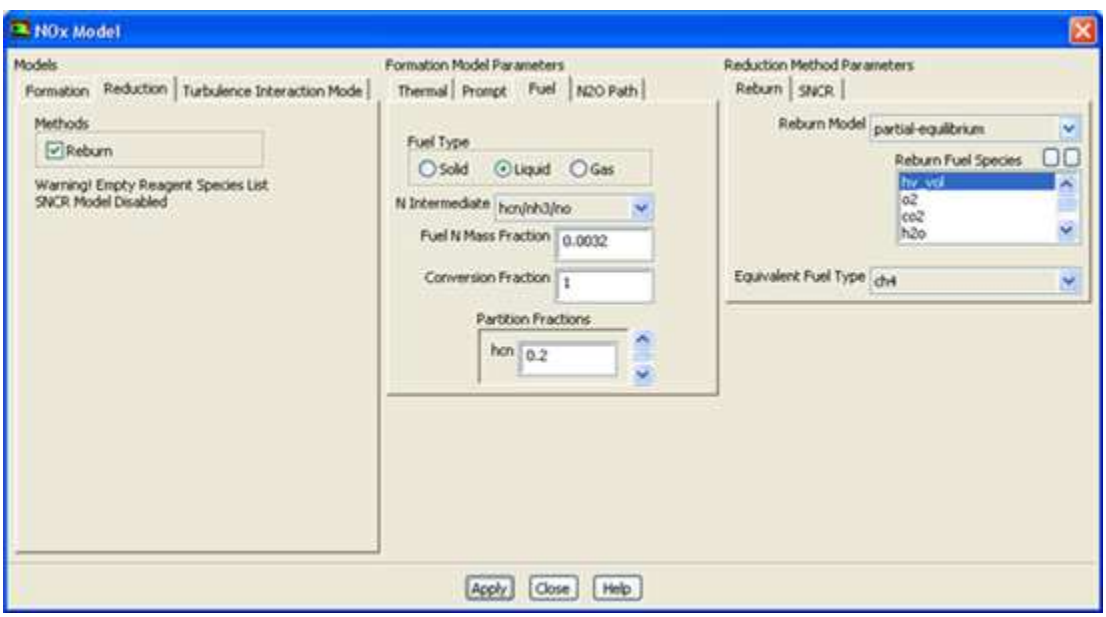
- To choose the instantaneous method, select **instantaneous [CH]** in the **Reburn Model** drop-down list.

  Important

  When you use this method, you must be sure to include the species CH, CH_2, and CH_3 in your problem definition. See NOx Reduction by Reburning in the Theory Guide for details.

- To choose the partial equilibrium method, select **partial-equilibrium** in the **Reburn Model** drop-down list. You will then need to select the **Reburn Fuel Species** from the list of available species. ANSYS FLUENT will allow you select up to 5 reburn fuel species. Specify the **Equivalent Fuel Type** (**ch4**, **ch3**, **ch2**, or **ch**). For example, if you choose methane as the reburn fuel, then the **Equivalent Fuel Type** would be **ch4**. If you choose a reburn fuel such as **hv_vol** (a volatile component of coal), then you must specify the most appropriate equivalent hydrocarbon fuel type so that the partial equilibrium model will be activated correctly.
- Due to coal volatiles behaving very differently, it is important to select the correct equivalent fuel type. You must first consider the volatile fuel composition, then check the C/H ratio to find the fuel which most closely matches CH, CH_2, CH_3, or CH_4 [*45*] (p. 2211). While the method for best determining the equivalent fuel is debatable, considering the C/H ratio of the fuel itself is a reasonable indicator.

Figure 23.3 The NOx Dialog Box Displaying the Reburn Reduction Method

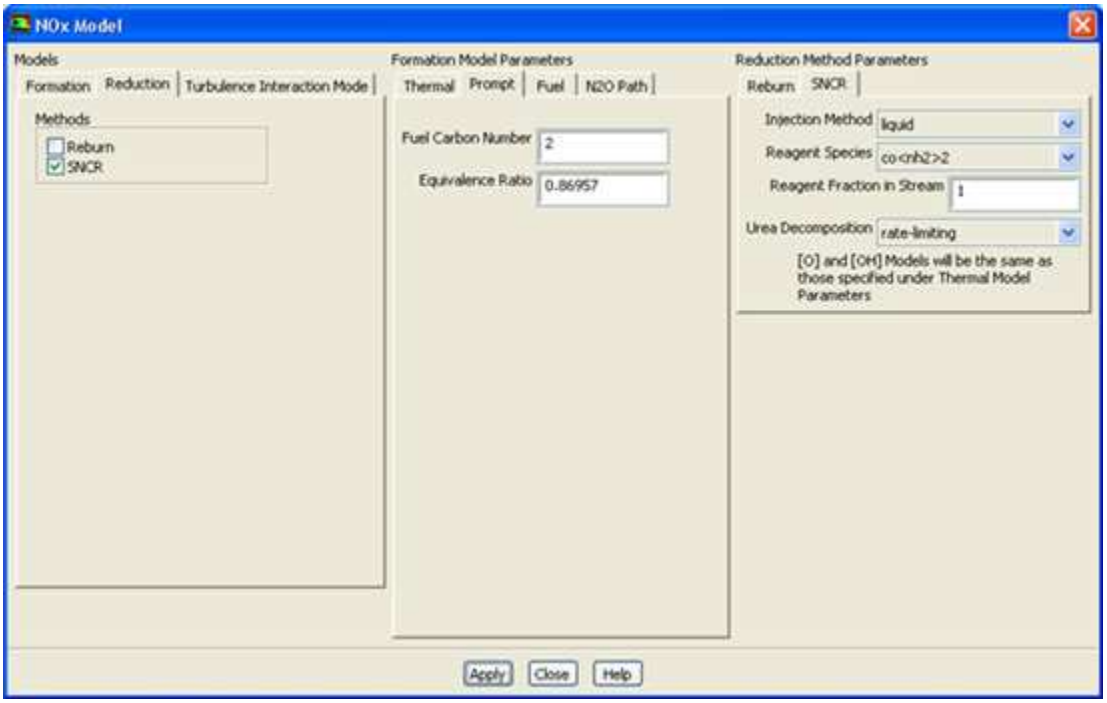


23.1.1.10. Setting SNCR Parameters

Prior to enabling reduction by SNCR, make sure that you have included in the species list **nh3** (for reduction by ammonia injection) and **co<nh2>2** (for reduction by urea injection). See NOx Reduction by SNCR in the Theory Guide for detailed information about SNCR theory.

To enable NO_x reduction by SNCR, click the **Reduction** tab in the *NOx Model Dialog Box* (p. 1706) and enable the **SNCR** option under **Methods**, as shown in *Figure 23.4* (p. 954).

Figure 23.4 The NOx Dialog Box Displaying the SNCR Reduction Method



Then click the **SNCR** tab under **Reduction Method Parameters**, where you can choose from the following options:

- To have ammonia or urea included as a gas-phase pollutant species from the injection locations, select **gaseous** in the **Injection Method** drop-down list.

  If you plan to select this option for NO_x postprocessing, then you must also include ammonia or urea as a gas-phase species. Additionally, you will need to specify the mass fraction of ammonia or urea at the respective inlet for the SNCR injection. You must include this set of inputs prior to the main ANSYS FLUENT combustion calculation.

- To have ammonia included as a liquid-phase pollutant species from the injection locations, select **liquid** in the **Injection Method** drop-down list. If urea is injected as a liquid solution, then select **liquid** in the **Injection Method** drop-down list. Note that you must activate the DPM model with urea or ammonia included as a material.

  If you plan to select this option for NO_x postprocessing, then you must include NH_3 as both a gas-phase and liquid-phase species. Additionally, you will need to specify injection locations for liquid droplet ammonia particles and set gaseous ammonia as the evaporation species. You need to include this set of inputs in conjunction with the main ANSYS FLUENT combustion calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Since urea is a subliming solid, and usually is injected as a solution, mixed in water, you have to define solid properties for urea under the *Create/Edit Materials Dialog Box* (p. 1736). It is assumed that the water evaporates before urea begins its subliming process. The sublimation process is modeled similar to the single rate devolatilization model of coal. You will supply the value for the sublimation rate (s^{-1}). You must specify the water content while defining the injection properties.

- Specify the SNCR **Reagent Species** as **nh3** (ammonia) or **co<nh2>2** (urea) in the drop-down list.
- When using the non-premixed combustion model with a liquid-phase reagent injection, enter a value in the **Reagent Fraction in Stream** to specify the mass fraction of the reagent in the reagent stream. The remaining mass fraction is assumed to be water. If you enabled a secondary stream in your PDF calculation, by default the secondary stream will act as the reagent stream. You can assign the primary stream as the reagent stream by using the text command that follows (enter `0` in response to the PDF `Stream ID` prompt that follows the `Injection Method` prompt):

  ```
  define/models/nox-parameters/nox-chemistry
  ```

- If the **Reagent Species** selected is **co <nh2> 2**, then you will either accept the **rate-limiting** option for **Urea Decomposition**, or specify the **NH3 Conversion** value when selecting a **user-specified Urea Decomposition**.

  You will use the urea decomposition under the **SNCR** tab to define which of the two decomposition models is to be used. The first model (which is the default) is the rate-limiting decomposition model, as given in Table 14.3: "Two-Step Urea Breakdown Process" in the Theory Guide. ANSYS FLUENT will then calculate the source terms according to the rates given in Table 14.3: "Two-Step Urea Breakdown Process" in the Theory Guide. The second model is for when you assume that the urea decomposes instantly into ammonia and HNCO at a given proportion. In this case, you will specify the molar conversion fraction for ammonia, assuming that the rest of the urea is converted to HNCO. An example value is given above.

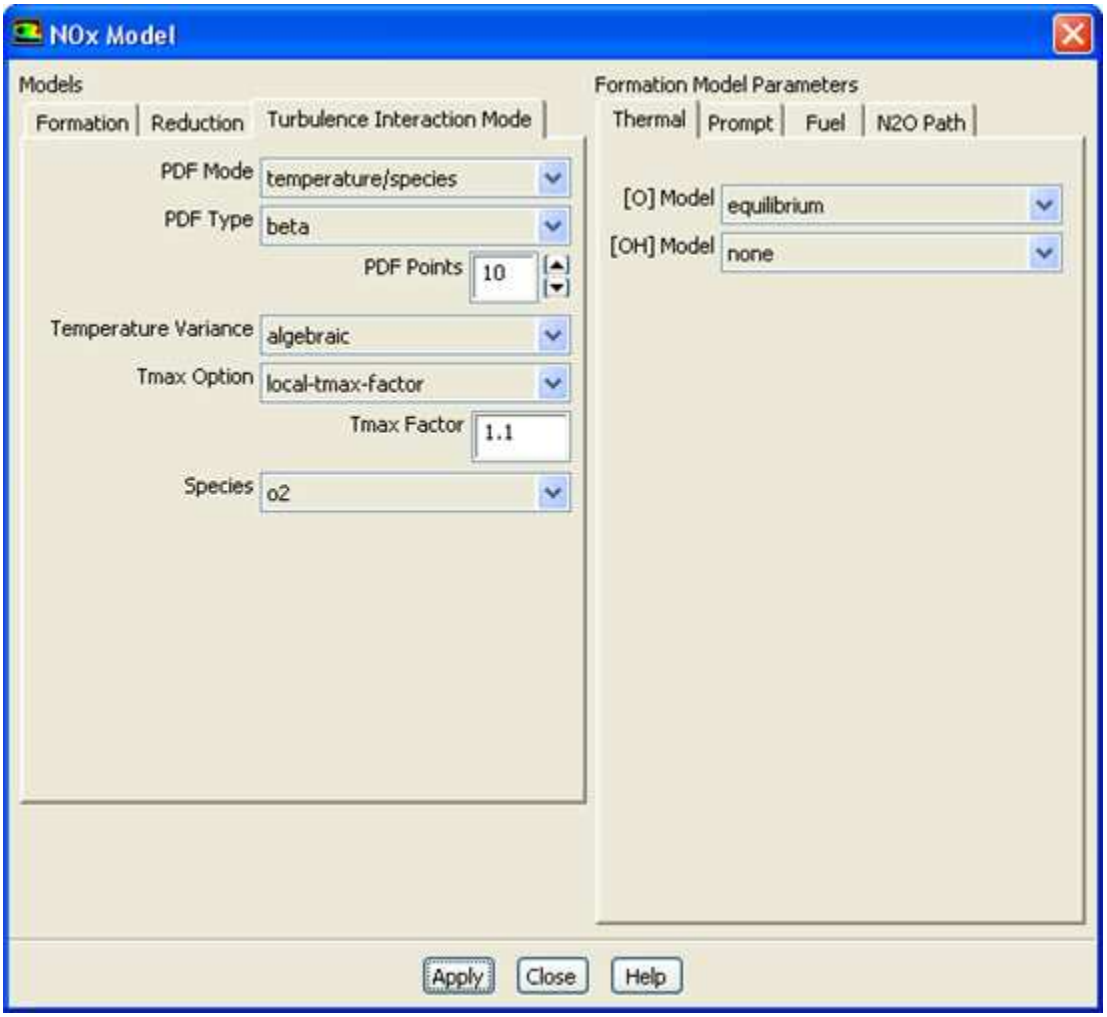
  The value for **user-specified** $\mathrm{NH_3}$ conversion is the mole fraction of $\mathrm{NH_3}$ in the mixture of $\mathrm{NH_3}$ and HNCO that is instantly created from the reagent injection. In this case, there is no urea source because all of reagent is assumed to convert to both $\mathrm{NH_3}$ and HNCO, instantly.

23.1.1.11. Setting Turbulence Parameters

If you want to take into account turbulent fluctuations (as described in NOx Formation in Turbulent Flows in the Theory Guide) when you compute the specified NO_x formation (thermal, prompt, and/or fuel, with or without reburn), define the turbulence parameters in the **Turbulence Interaction Mode** tab.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.5 The NOx Model Dialog Box and the Turbulence Interaction Mode Tab



Select one of the options in the **PDF Mode** drop-down list in the **Turbulence Interaction Mode** tab:

- Select **temperature** to take into account fluctuations of temperature.
- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s).

Important

When modeling the formation of other pollutants along with NO_x, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** group boxes of the *NOx Model Dialog Box* (p. 1706) and the **Turbulence Interaction Mode** group boxes of the **SOx Model** and **Soot Model** dialog boxes. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fraction and enthalpy and the instantaneous NO_x rates are calculated at each cell. The PDF used for convoluting the instantaneous NO_x rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF for mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous NO_x rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** tab:

PDF Type

allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 14–108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 14–111 in the Theory Guide.

PDF Points

allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 14–105 or Equation 14–106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (e.g., 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 14–114 of the Theory Guide). You have the option of selecting **transported** to instead solve Equation 14–113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option

provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature:

- The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field.
- You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.
- You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**.
- If you have selected a user-defined function in the **NOx Rate** drop-down list in the **Formation** tab, then you can select **user-defined** so that the temperature limit is specified by a UDF. See the UDF Manual for details about user-defined functions.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the NO_x formation calculations.

Important

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 14–114 in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.1.1.12. Defining Boundary Conditions for the NOx Model

At flow inlet boundaries, you will need to specify the **Pollutant NO Mass Fraction**, and if necessary, the **Pollutant HCN Mass Fraction**, **Pollutant NH3 Mass Fraction**, and **Pollutant N2O Mass Fraction**.

Boundary Conditions

You can retain the default inlet values of zero for these quantities or you can input nonzero numbers as appropriate for your combustion system.

23.1.2. Solution Strategies

To solve for NO_x products, perform the following steps:

- (optional) If the discrete phase model (DPM) is activated (by turning on the **Interaction with Continuous Phase**) to run with the NO_x model, then set the **Number of Continuous Phase Iterations per DPM Iteration** to `0` such that no DPM iterations are performed as the NO_x case is being solved.
- In the *Equations Dialog Box* (p. 1904) of the *Solution Controls Task Page* (p. 1902), turn off the solution of all variables except species NO (and HCN, NH_3, or N_2O, based on the model selected).

  Solution Controls → **Equations...**

- Also in the **Solution Controls** task page, set a suitable value for the NO (and HCN, NH_3, or N_2O, if appropriate) under-relaxation. A value of 0.95 is suggested, although lower values may be required for certain problems (i.e., if convergence cannot be obtained, try a lower under-relaxation value).
- In the *Residual Monitors Dialog Box* (p. 1915), decrease the convergence criterion for NO (and HCN, NH_3, or N_2O, if appropriate) to 10^{-6}.

  Monitors → **Residuals** → **Edit...**

- Perform calculations until convergence (i.e., until the NO—and HCN, NH_3, or N_2O, if solved—species residuals are below 10^{-6}) to ensure that the NO and HCN or NH_3 concentration fields are no longer evolving.

  Run Calculation

23.1.3. Postprocessing

When you compute NO_x formation, the following additional variables will be available for postprocessing:

- **Mass fraction of Pollutant no**
- **Mass fraction of Pollutant hcn** (appropriate fuel NO_x model only)
- **Mass fraction of Pollutant nh3** (appropriate fuel NO_x model only)
- **Mass fraction of Pollutant n2o** (N_2O-intermediate model only)
- **Mass fraction of Pollutant urea** (SNCR urea injection)
- **Mass fraction of Pollutant hnco** (SNCR urea injection)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mass fraction of Pollutant nco** (SNCR urea injection)
- **Mole fraction of Pollutant no**
- **Mole fraction of Pollutant hcn** (appropriate fuel NO_x model only)
- **Mole fraction of Pollutant nh3** (appropriate fuel NO_x model only)
- **Mole fraction of Pollutant n2o** (N_2O-intermediate model only)
- **Mole fraction of Pollutant urea** (SNCR urea injection)
- **Mole fraction of Pollutant hnco** (SNCR urea injection)
- **Mole fraction of Pollutant nco** (SNCR urea injection)
- **no Density**
- **hcn Density** (appropriate fuel NO_x model only)
- **nh3 Density** (appropriate fuel NO_x model only)
- **n2o Density** (N_2O-intermediate model only)
- **urea Density** (SNCR urea injection)
- **hnco Density** (SNCR urea injection)
- **nco Density** (SNCR urea injection)
- **Rate of no** (from the individual pathways)
- **Rate of hcn** (appropriate fuel NO_x model only)
- **Rate of nh3** (appropriate fuel NO_x model only)
- **Rate of n2o** (N_2O-intermediate model only)
- **Rate of urea** (SNCR urea injection)
- **Rate of hnco** (SNCR urea injection)
- **Rate of nco** (SNCR urea injection)

These variables are contained in the **NOx...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Additional NO rates from individual pathways, **Thermal**, **Prompt**, **Fuel**, **N2O Path**, and **SNCR** can be plotted.

23.2. SOx Formation

The following sections describe how to use the SO_x model in ANSYS FLUENT. For information about the theory behind the SO_x models in ANSYS FLUENT, see SOx Formation in the Theory Guide.

23.2.1. Using the SOx Model
23.2.2. Solution Strategies
23.2.3. Postprocessing

23.2.1. Using the SOx Model

When the sulfur content in the fuel is low, SO_x concentrations that are generated in combustion generally have minimal influence on the predicted flow field, temperature, and major combustion product concentrations. The most efficient way to use the SO_x model is as a postprocessor to the main combustion calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, if the sulfur content is high, then SO_x formation should be coupled with the gas phase combustion process rather than treating it as a postprocessing step.

The procedure for activating and setting up the model for a decoupled solution is as follows:

1. Calculate your combustion problem using ANSYS FLUENT.

   Important

   The premixed combustion model is not compatible with the SO_x model.

2. Enable the SO_x model, define the fuel streams, and set the appropriate parameters, as described in this section.

   Models → **SOx** → **Edit...**

3. Define the boundary conditions for SO_2 and H_2S (and SO_3, SH, or SO if necessary) at flow inlets.

   Boundary Conditions

4. In the *Equations Dialog Box* (p. 1904), turn off the solution of all variables except species SO_2 and H_2S (and SO_3, SH, or SO, based on your selections).

   Solution Controls → **Equations...**

5. Perform calculations until convergence (i.e., until the SO_2 and H_2S—and SO_3, SH, or SO, if solved—species residuals are below 10^{-6}) to ensure that the SO_2 and H_2S concentration fields are no longer evolving.

   Run Calculation

6. Review the mass fractions of SO_2 and H_2S (and SO_3, SH, or SO) by generating graphical plots or alphanumeric reports in the usual way.
7. Save a new set of case and data files, if desired.

   File → **Write** → **Case & Data...**

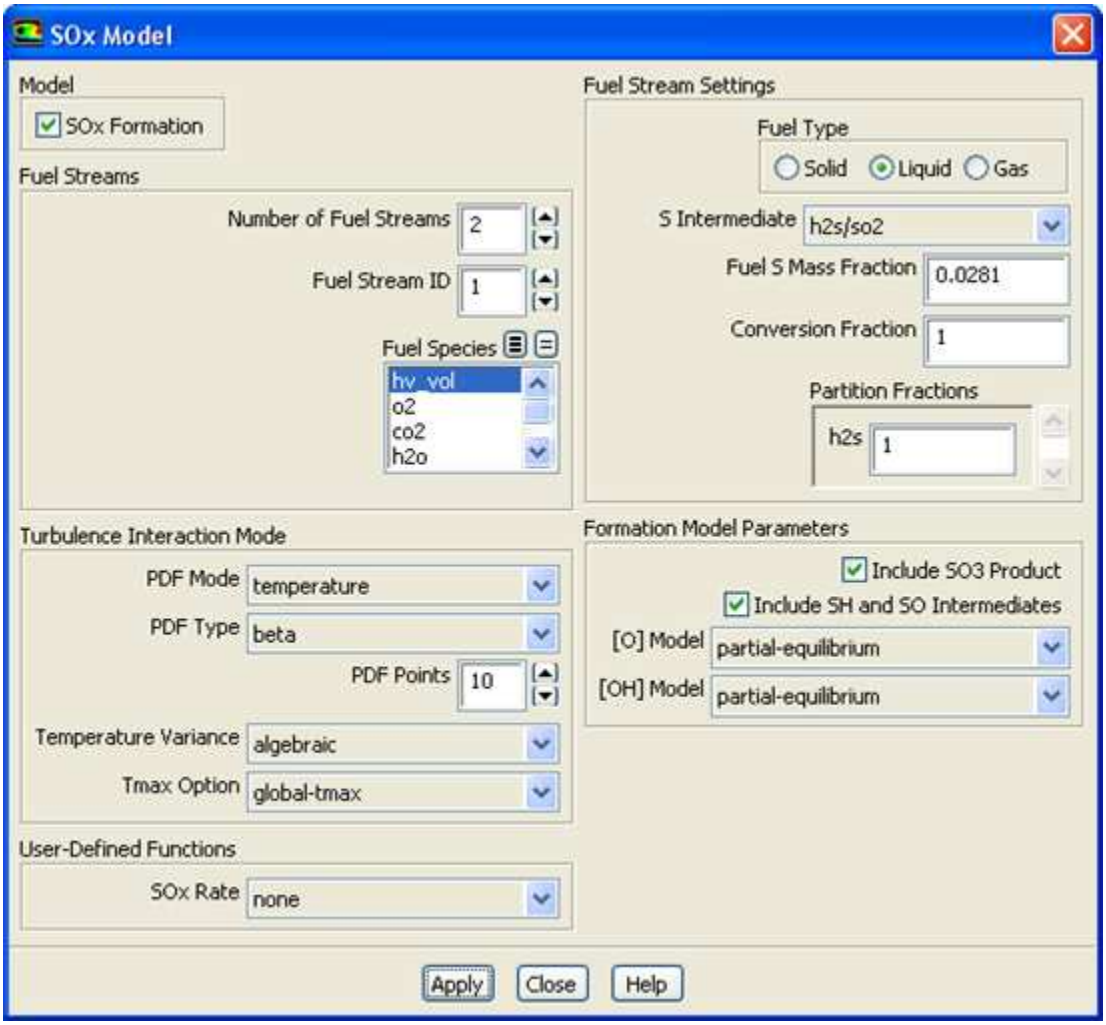
23.2.1.1. Enabling the SOx Model

To model SO_x formation, enable the **SOx Formation** option in the *SOx Model Dialog Box* (p. 1713) (*Figure 23.6* (p. 961)).

Models → **SOx** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.6 The SOx Model Dialog Box



23.2.1.2. Defining the Fuel Streams

ANSYS FLUENT allows you to define multiple fuel streams when you are modeling SO_x formation, as shown in the following steps:

1. Specify the **Number of Fuel Streams** in the **Fuel Streams** group box. You are allowed up to three separate fuel streams.
2. Define the first fuel stream.
   a. Select the fuel stream to be defined by using the arrow keys of the **Fuel Stream ID** text box.
   b. When the non-premixed combustion model is not enabled, select the fuel species from the **Fuel Species** list. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10.
   c. When the non-premixed combustion model is enabled (*Figure 23.7* (p. 962)), make a selection from the **PDF Stream** drop-down list to define the species for this stream. You can select either the **primary** or **secondary** fuel stream species, as defined in the PDF table.
   d. Specify the parameters for this particular fuel stream in the **Fuel Stream Settings** group box. See *Defining the SOx Fuel Stream Settings* (p. 963) for details.
3. Repeat steps 2.(a)–2.(d) for each additional fuel stream.

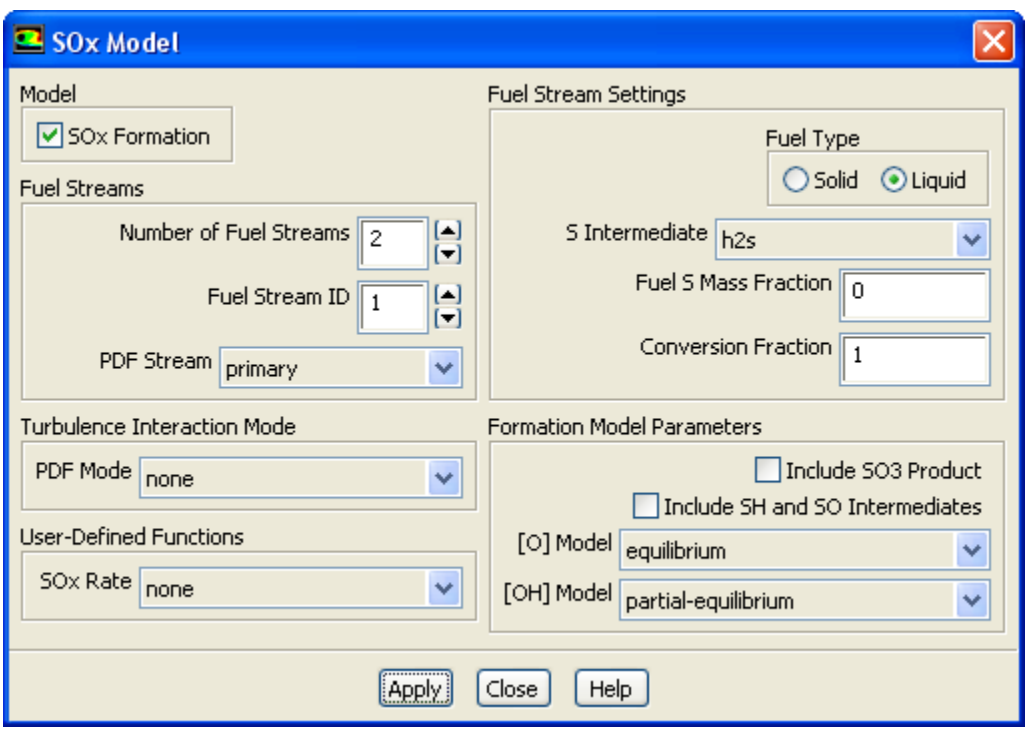
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set the formation model parameters that apply to all of the fuel streams in the **Formation Model Parameters** group box:
   - You have the option of including SO_3 as a product, and including SH and SO as intermediates by enabling the **Include SO3 Product** and the **Include SH and SO Intermediates** options, respectively. See Reaction Mechanisms for Sulfur Oxidation in the Theory Guide for further information.
   - Specify the method by which O and OH will be calculated. The SO_x routines employ three methods for reduction calculations of SO_x:
     - You can select **equilibrium**, **partial-equilibrium**, or **instantaneous** in the **[O] Model** drop-down list.
     - You can select **none**, **partial-equilibrium**, or **instantaneous** in the **[OH] Model** drop-down list.

Important

To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

Figure 23.7 The SOx Model Dialog Box with Non-Premixed Combustion



Note that the following limitations apply when you are modeling SO_x formation with multiple fuel streams, if more than one fuel stream has the same fuel type (as defined in the **Fuel Type** list in the **Fuel Stream Settings** group box):

- You should only use multiple liquid fuel streams or multiple solid (coal) fuel streams when you have streams that have different destination species. Therefore, you must make sure that the injectors associated with the fuel streams specify these unique destination species correctly, as defined in the **Devo-**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

latilizing Species drop-down list in the *Set Injection Properties Dialog Box* (p. 2097) (see *Defining Injection Properties* (p. 1040) for details).

- For multiple solid (coal) fuel streams, the fuel streams should have the same char-related parameter values in the **Fuel Stream Settings** group box — i.e., the **Char S Mass Fraction** and the **Partition Fractions** (for char S) values. Note that even if different values are set for these char-related parameters, ANSYS FLUENT will only recognize those specified for the solid fuel stream with the lowest ID number, and then apply them to all of the other solid fuel streams.

For more information about the limitations associated with multiple fuel streams with the same fuel type, contact your ANSYS FLUENT support engineer.

Important

Note that if you read a case file with SO_x settings that was set up in a version of ANSYS FLUENT previous to 12, you must make a selection for the fuel species. This selection should be made either in the **PDF Stream** drop-down list for non-premixed combustion, or in the **Fuel Species** list for all other combustion models.

23.2.1.3. Defining the SOx Fuel Stream Settings

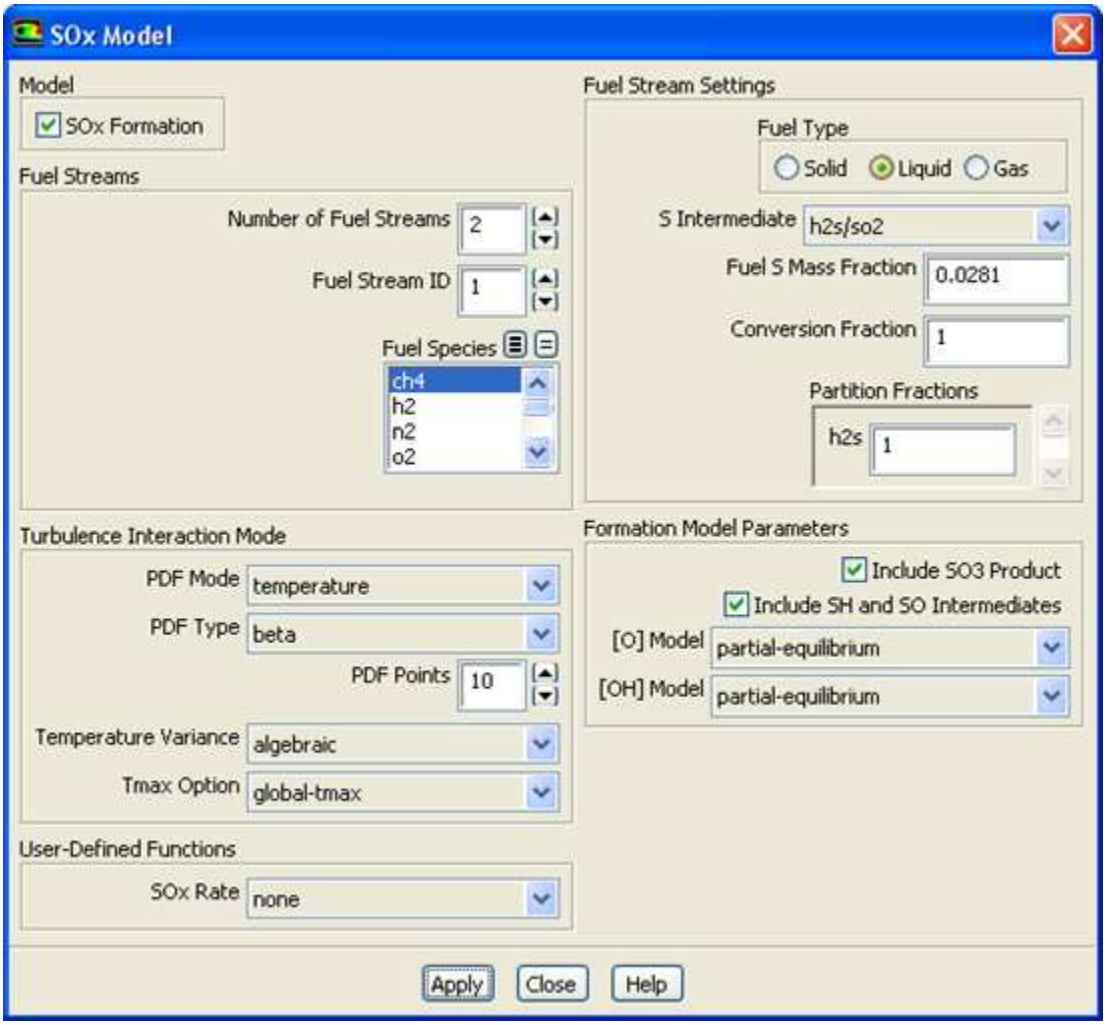
When using the SO_x model, you must set the parameters in the **Fuel Stream Settings** group box for each fuel stream specified in the **Fuel Stream ID** text box.

To begin, specify the fuel type in the following manner:

- To calculate SO_x formation from a solid fuel, select **Solid** under **Fuel Type**.
- To calculate SO_x formation from a liquid fuel, select **Liquid** under **Fuel Type**.
- To calculate SO_x formation from a gaseous fuel, select **Gas** under **Fuel Type**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.8 The SOx Model Dialog Box Displaying Liquid Fuel Parameters



Note that you can use only one of the fuel types for a given fuel stream. The **Gas** option is available only when the **Species Transport** model is enabled (see *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801)).

23.2.1.3.1. Setting SOx Parameters for Gaseous and Liquid Fuel Types

If you have selected **Gas** or **Liquid** as the **Fuel Type**, you will also need to specify the following:

- Select the intermediate species (**h2s**, **so2**, or **h2s/so2**) in the **S Intermediate** drop-down list.
- Set the correct mass fraction of sulfur in the fuel (kg sulfur per kg fuel) in the **Fuel S Mass Fraction** field.
- Specify the overall fraction of the fuel S, by mass, that will be converted to the intermediate species and/or product SO_2 in the **Conversion Fraction** field. Thus, any remaining S will not contribute to SO_x formation. This is based on the assumption that the remaining volatile S will convert to gas phase sulfur. The **Conversion Fraction** for the **S Intermediate** has a default value of 1.
- If you selected **h2s/so2** as the intermediate, you will need to set the fraction of the converted fuel S, by mass, that will become **h2s** under **Partition Fractions**. The fraction of fuel S that will become SO_2 will be calculated by the remainder.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that setting a partition fraction of 0 for H_2S is equivalent to assuming that all fuel S is converted to the final product SO_2.

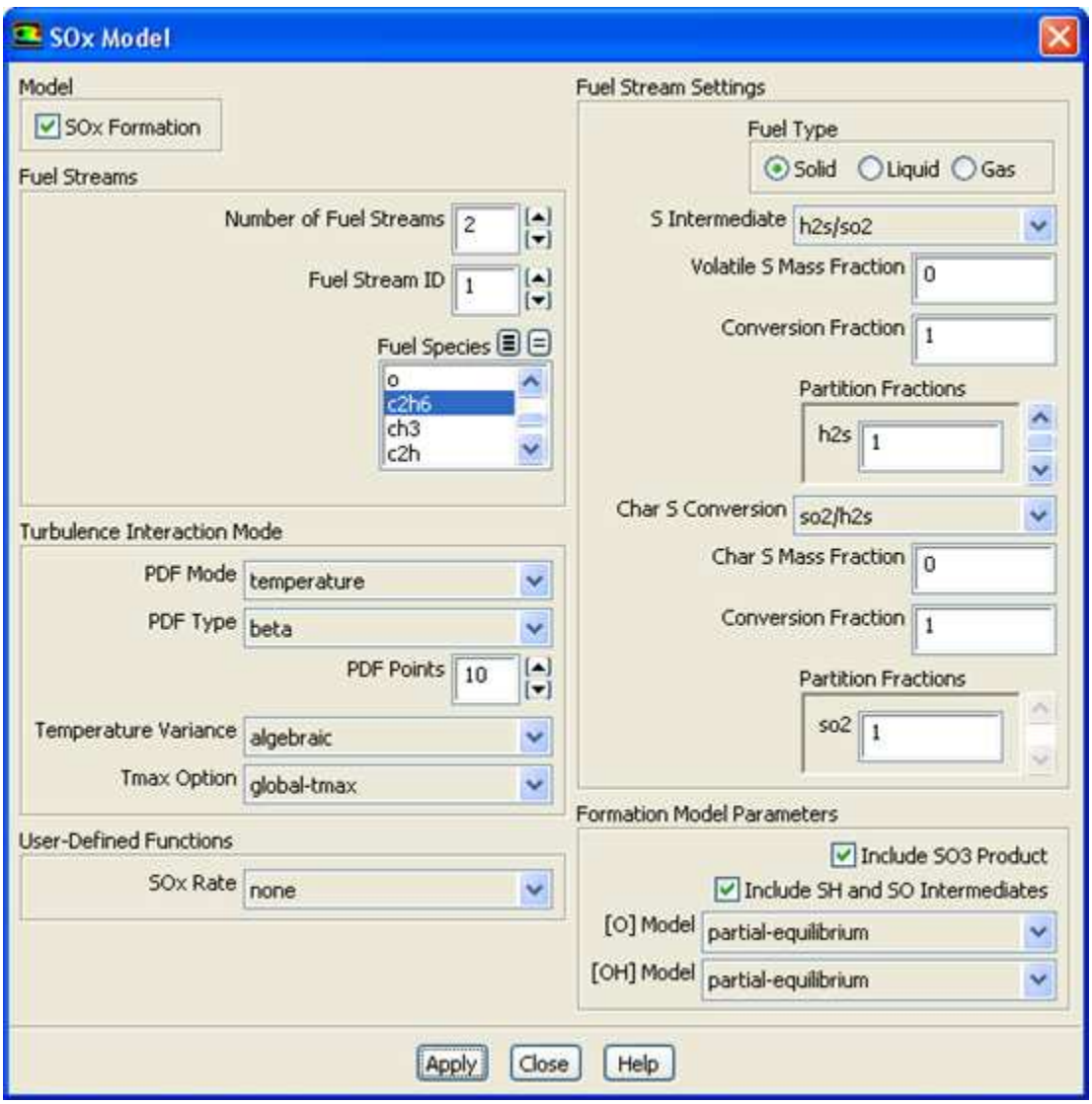
Important

Note that there is a limitation that must be considered when defining more than one liquid fuel stream. See *Defining the Fuel Streams* (p. 946) for details.

23.2.1.3.2. Setting SOx Parameters for a Solid Fuel

For solid fuel, several inputs are required for the SO_x model.

Figure 23.9 The SOx Model Dialog Box Displaying Solid Fuel Parameters



- Select the intermediate species (**h2s**, **so2**, or **h2s/so2**) in the **S Intermediate** drop-down list.
- Specify the mass fraction of sulfur in the volatiles in the **Volatile S Mass Fraction** field.
- Specify the overall fraction of the volatile S, by mass, that will be converted to the intermediate species and/or product SO_2 in the **Conversion Fraction** field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you selected **h2s/so2** as the volatile S intermediate, you will need to specify the fraction of the converted volatile S, by mass, that will become **h2s** under **Partition Fractions**. The fraction of volatile S that will become SO_2 will be calculated by the remainder.
- Select the char S conversion path from the **Char S Conversion** drop-down list as **so2**, **h2s**, or **so2/h2s**.
- Specify the mass fraction of sulfur in the char in the **Char S Mass Fraction** field.
- Specify the overall fraction of the char S, by mass, that will be converted to the intermediate species and/or product SO_2 in the **Conversion Fraction** field.
- If you selected **so2/h2s** from the **Char S Conversion** drop-down list, you will need to specify the fraction of the converted char S, by mass, that will become SO_2 under **Partition Fractions**. The fraction of char S that will become H_2S will be calculated by the remainder.

Important

Note that there are limitations that must be considered when defining more than one solid fuel stream. See *Defining the Fuel Streams* (p. 961) for details.

The following equations are used to determine the mass fraction of sulfur in the volatiles and char:

$$\dot{m}_{S_{v/c}} = \dot{m}_{v/c} * mf_{S_{v/c}} \tag{23–8}$$

where

$\dot{m}_{S_{v/c}}$ = rate of release of fuel sulfur in kg/s

$\dot{m}_{v/c}$ = rate of release of volatiles (v) or char (c) in kg/s

$mf_{S_{v/c}}$ = mass fraction of sulfur in volatiles or char

Let

TS_{fuel} = total sulfur mass fraction in daf coal (i.e., from daf ultimate analysis)

S_{split} = char sulfur as a fraction of total sulfur

F_{vol} = mass fraction of volatiles in daf coal

F_{char} = mass fraction of char in daf coal

Then the following should hold:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$F_{vol}+F_{char}=1 \tag{23–9}$$

$$\frac{F_{char}*mf_{S_c}}{TS_{fuel}}=S_{split} \tag{23–10}$$

$$F_{vol}*mf_{S_v}+F_{char}*mf_{S_c}=TS_{fuel} \tag{23–11}$$

$$mf_{S_v}=\left(1-S_{split}\right)*\frac{TS_{fuel}}{F_{vol}} \tag{23–12}$$

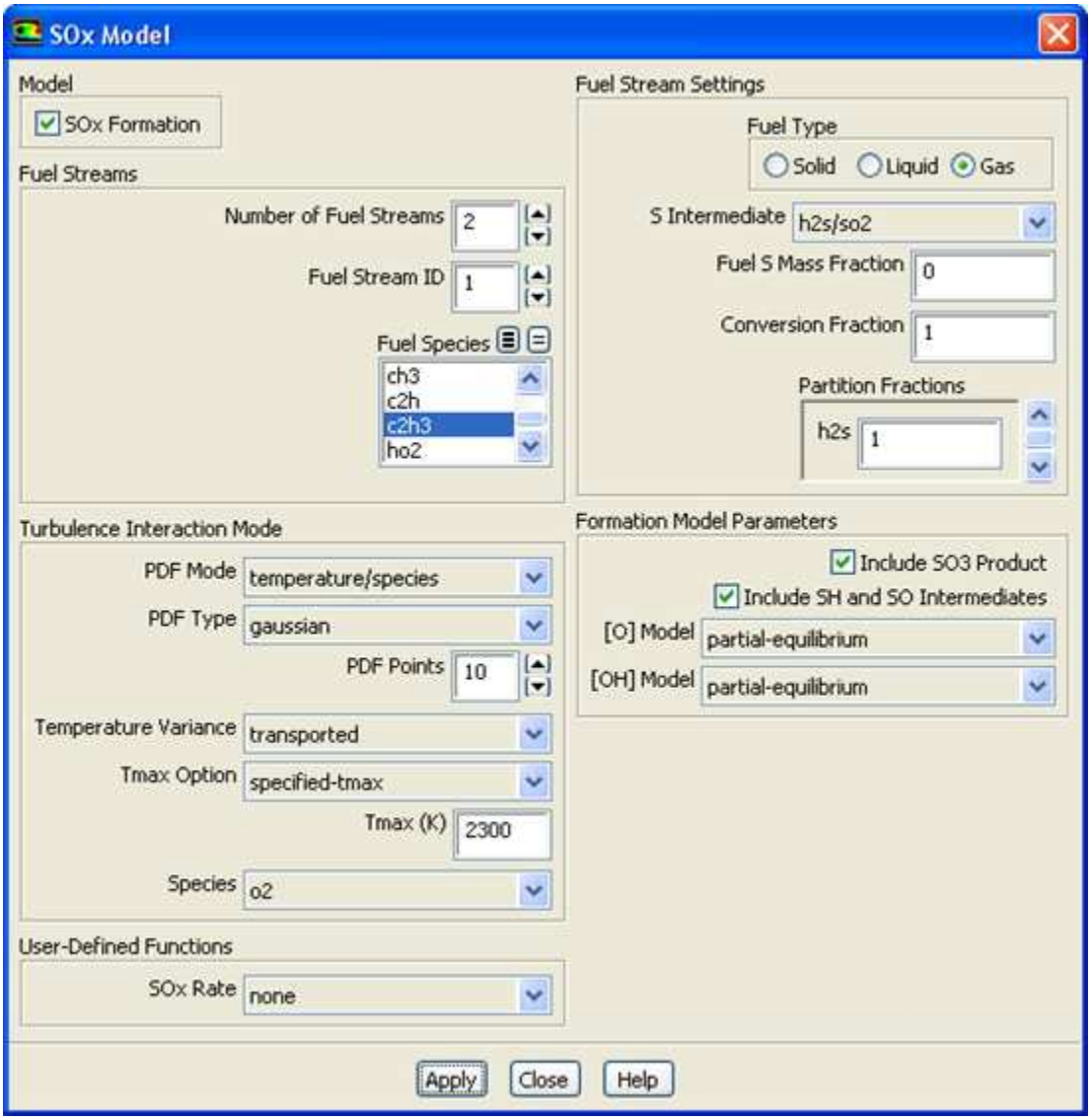
$$mf_{S_c}=S_{split}*\frac{TS_{fuel}}{F_{char}} \tag{23–13}$$

23.2.1.4. Setting Turbulence Parameters

If you want to take into account turbulent fluctuations when you compute the specified SO_2 formation, define the turbulence parameters in the **Turbulence Interaction Mode** group box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.10 The SOx Model Dialog Box for a Gas Fuel Type with Turbulence



Select one of the options in the **PDF Mode** drop-down list:

- Select **temperature** to take into account fluctuations of temperature.
- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s).

Important

When modeling the formation of other pollutants along with SO_x, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** tab of the *NOx Model Dialog Box* (p. 1706) and the **Turbulence Interaction Mode** group boxes of the **SOx Model** and **Soot Model** dialog boxes. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous SO_x rates are calculated at each cell. The PDF used for convoluting the instantaneous SO_x rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF for mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous SO_x rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** group box:

PDF Type

allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 14–108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 14–111 in the Theory Guide.

PDF Points

allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 14–105 or Equation 14–106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (e.g., 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 14–114 in the Theory Guide). You have the option of selecting **transported** to instead solve Equation 14–113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option

provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature:

- The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field.
- You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.
- You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**.
- If you have selected a user-defined function from the **SOx Rate** drop-down menuin the **User-Defined Functions** group box, then you can select **user-defined** so that the limit is specified by a UDF. See the UDF Manual for details about user-defined functions.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the SO_x formation calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 14–114 in the Theory Guide).

23.2.1.5. Specifying a User-Defined Function for the SOx Rate

You can choose to specify a user-defined function for the rate of SO_x production. By default, the rate returned from the UDF is added to the rate returned from the standard SO_x production options. You also have the option of replacing ANSYS FLUENT's SO_x rate calculations with your own user-defined SO_x rate.

In addition to or instead of using the UDF to specify the SO_x rate, you can use it to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

To use a UDF to add a rate to ANSYS FLUENT's SO_x rate calculations, you must compile and load the desired function, and then select it from the **SOx Rate** drop-down list in the **User-Defined Functions** group box. After you have selected the UDF, you have the following options:

- You can specify that your custom rate is added to the ANSYS FLUENT SO_x rate calculations, by retaining the default selection of **Add to the FLUENT Rate** in the **UDF Rate** group box.
- You can replace the ANSYS FLUENT SO_x rate calculations with your custom rate, by selecting **Replace FLUENT Rate** in the **UDF Rate** group box.
- You can specify custom values for T_{max}, by selecting **user-defined** from the **Tmax Option** drop-down list in the **Turbulence Interaction Mode** group box.

See the UDF Manual for details about user-defined functions.

23.2.1.6. Defining Boundary Conditions for the SOx Model

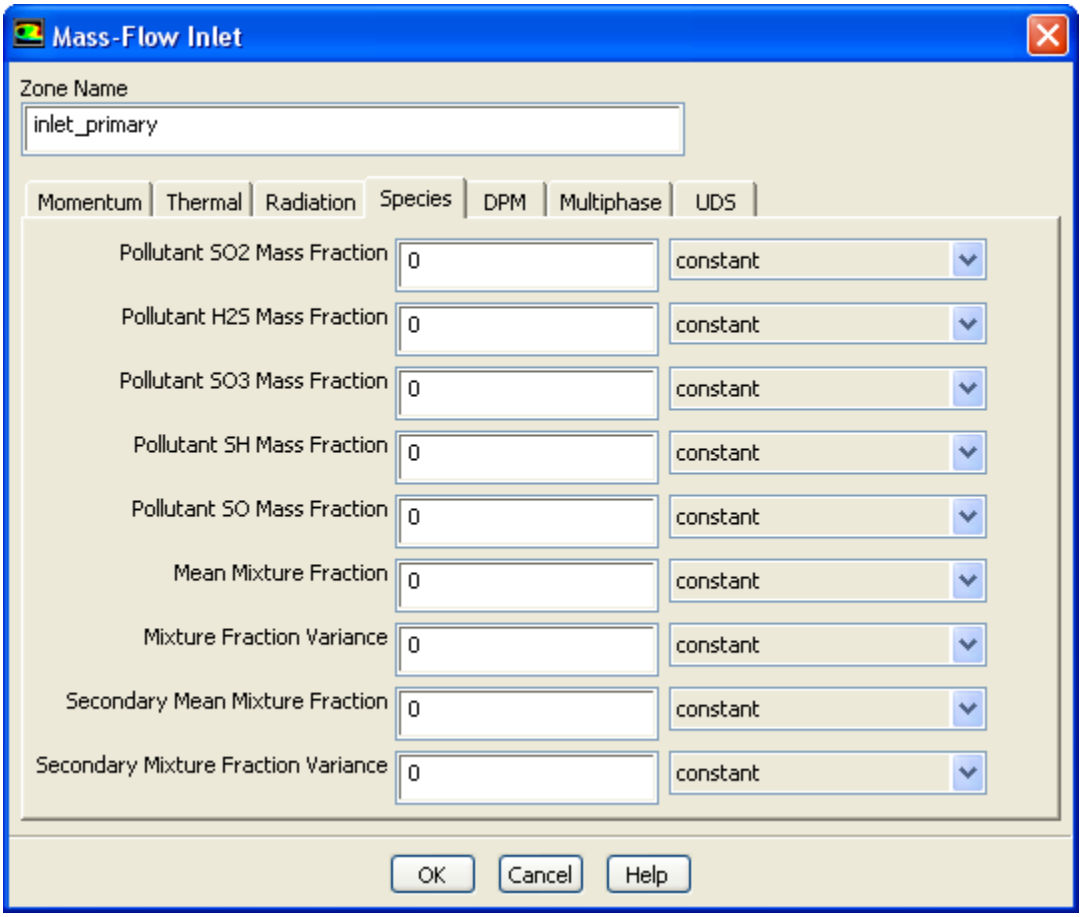
At flow inlet boundaries, you will need to specify the **Pollutant SO Mass Fraction**, and if necessary, the **Pollutant SH Mass Fraction**, **Pollutant H2S Mass Fraction**, **Pollutant SO3 Mass Fraction**, and **Pollutant SO2 Mass Fraction** in the **Species** tab, as demonstrated in *Figure 23.11* (p. 971).

Boundary Conditions

You can retain the default inlet values of zero for these quantities or you can input nonzero numbers as appropriate for your combustion system.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.11 The Mass-Flow Inlet Dialog Box and the Species Tab



23.2.2. Solution Strategies

To solve for SO_x products:

- (optional) If the discrete phase model (DPM) is activated (by turning on the **Interaction with Continuous Phase**) to run with the SO_x model, then set the **Number of Continuous Phase Iterations per DPM Iteration** to `0` such that no DPM iterations are performed as the SO_x case is being solved.
- In the *Equations Dialog Box* (p. 1904), turn off the solution of all variables except **Pollutant so2** and **Pollutanth2s** (and **Pollutant so3**, **Pollutant sh**, and **Pollutant so**, if applicable).

  Solution Controls → **Equations...**

- Also in the *Solution Controls Task Page* (p. 1902), set suitable values for the pollutant SO_2 and H_2S (and SO_3, SH, and SO, if applicable) under-relaxation factors. A value of 0.95 is suggested, although lower values may be required for certain problems. That is, if convergence cannot be obtained, try a lower under-relaxation value.

  Solution Controls

- Under **Spatial Discretization** in the *Solution Methods Task Page* (p. 1900), select the desired scheme for each of the pollutants, SO_2 and H_2S (and SO_3, SH, and SO, if applicable)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Methods

- In the *Residual Monitors Dialog Box* (p. 1915), decrease the convergence criterion for the pollutants SO_2 and H_2S (and SO_3, SH, and SO, if applicable) to 10^{-6}.

Solve $\rightarrow$ **Monitors** $\rightarrow$ **Residual...**

- Perform calculations until convergence (i.e., until the SO_2 and H_2S—and SO_3, SH, and SO—pollutant residuals are below 10^{-6}) to ensure that the SO_2 and H_2S (and SO_3, SH, and SO) concentration fields are no longer evolving.

Run Calculation

- Review the mass fractions of pollutants, SO_2 and H_2S (and SO_3, SH, and SO), by generating graphical plots or alphanumeric reports as described in *Postprocessing* (p. 972).

23.2.3. Postprocessing

When you compute SO_x formation, the following additional variables will be available for postprocessing. They are contained in the **SOx...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

- **Mass fraction of pollutant so2**
- **Mass fraction of pollutant h2s**
- **Mass fraction of pollutant so3**
- **Mass fraction of pollutant sh**
- **Mass fraction of pollutant so**
- **Mole fraction of pollutant so2**
- **Mole fraction of pollutant h2s**
- **Mole fraction of pollutant so3**
- **Mole fraction of pollutant sh**
- **Mole fraction of pollutant so**
- **so2 Density**
- **h2s Density**
- **so3 Density**
- **sh Density**
- **so Density**
- **Rate of so2**
- **Rate of h2s**
- **Rate of so3**
- **Rate of sh**
- **Rate of so**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.3. Soot Formation

This section contains information about using the soot formation models in ANSYS FLUENT. For information about the theory behind the soot models in ANSYS FLUENT, see Soot Formation in the Theory Guide.

23.3.1. Using the Soot Models

23.3.1. Using the Soot Models

When the mass fraction of soot is relatively large (e.g., 10%) or if your problem involves the effect of radiation, the soot formation should be computed as part of the main combustion solution and not through postprocessing (as is done for the NO_x and SO_x models). The procedure for setting up and solving a soot formation model is outlined below, and described in detail on the pages that follow. Remember that only the steps that are pertinent to soot modeling are shown here. For information about inputs related to other models that you are using in conjunction with the soot formation model, see the appropriate sections for those models.

1. Set up your combustion problem using ANSYS FLUENT as usual. Note the following limitations:
   - None of the soot models are compatible with premixed combustion.
   - Only the Moss-Brookes model and the Hall extension are compatible with non-premixed and partially premixed combustion.
   - The one-step and two-step soot formation models are only available for turbulent flows.
2. Enable the desired soot formation model and set the related parameters, as described in this section.

   Models → **Soot** → **Edit...**
3. Define the boundary conditions for soot (and nuclei, if you are not using the one-step model) at flow inlets.

   Boundary Conditions
4. In the *Solution Controls Task Page* (p. 1902), set a suitable value for the soot (and nuclei, if you are not using the one-step model) under-relaxation factor(s). The default value is 0.9. If convergence cannot be obtained with this value, try a lower under-relaxation value.

   Solution Controls
5. Perform calculations until convergence (i.e., until the soot / nuclei residual is below 10^{-6}) to ensure that the soot (and nuclei) field is no longer evolving.

   Run Calculation

   Note that when **Soot-Radiation Interaction** is enabled in the **Soot Model** dialog box, soot equations (soot, nuclei, and tar species, if available) must be solved together with the combustion solution to obtain correct radiation coupling. Therefore, when **Soot-Radiation Interaction** is enabled, the soot equations are not available for pollutant postprocessing, but will be solved together with the combustion solution. However, if other pollutants are to be solved (while soot/radiation interaction is enabled), those pollutant equations will be postprocessed if you so desire.
6. Review the mass fraction of soot (and nuclei) by generating graphical plots or alphanumeric reports in the usual way.
7. Save a new set of case and data files, if desired.

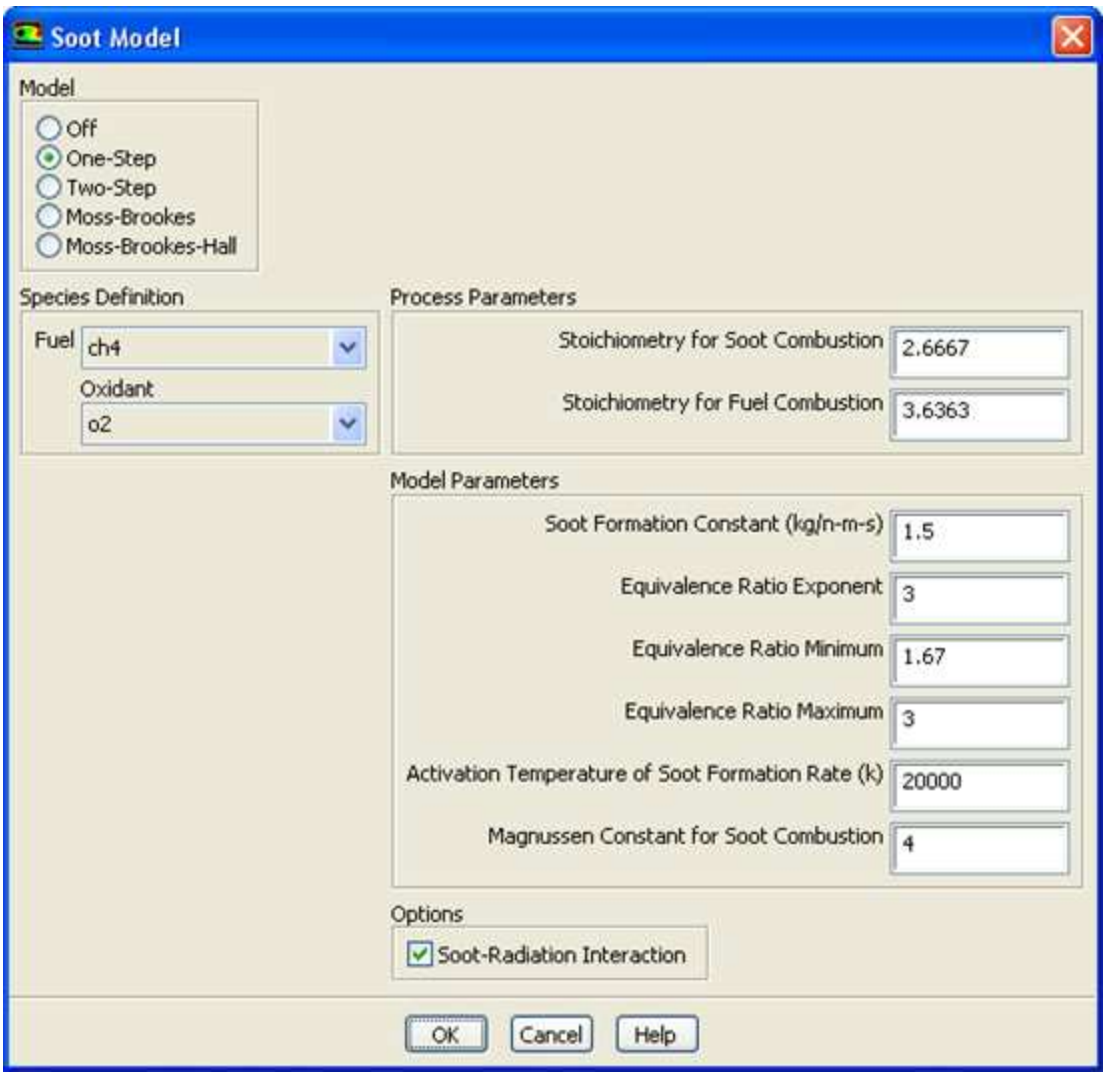
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.3.1.1. Setting Up the One-Step Model

You can enable and set up the one-step soot formation model by using the *Soot Model Dialog Box* (p. 1717) (*Figure 23.12* (p. 974)).

Models → **Soot** → **Edit...**

Figure 23.12 The Soot Model Dialog Box for the One-Step Model



Under **Model**, select **One-Step**. The dialog box will expand to show the appropriate inputs.

Next, you need to tell ANSYS FLUENT which chemical species in your model should be used as the fuel and oxidizer. Under **Species Definition**, select the fuel in the **Fuel** drop-down list and the oxidizer in the **Oxidant** drop-down list. If you are using the non-premixed model for the combustion calculation and your fuel stream consists of a mixture of components, you should choose the most appropriate species as the **Fuel** species for the soot formation model. Similarly, the most significant oxidizing component (e.g., O_2) should be selected as the **Oxidant**.

If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You must next define the **Process Parameters**, by inputting the stoichiometry of the fuel and soot combustion for the one-step model:

Stoichiometry for Soot Combustion
is the mass stoichiometry, v_{soot}, in Equation 14–131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS FLUENT (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion
is the mass stoichiometry, v_{fuel}, in Equation 14–131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS FLUENT (3.6363) is for combustion of propane (C_3H_8) by oxygen (O_2).

You must then set the **Modeling Parameters** that are used in Equation 14–128, Equation 14–130, and Equation 14–131 in the Theory Guide:

Soot Formation Constant
is the parameter C_s in Equation 14–128 in the Theory Guide.

Equivalence Ratio Exponent
is the exponent r in Equation 14–128 in the Theory Guide.

Equivalence Ratio Minimum
and **Equivalence Ratio Maximum** are the minimum and maximum values of the fuel equivalence ratio ϕ in Equation 14–128 in the Theory Guide. This equation will be solved only if **Equivalence Ratio Minimum** $< \phi <$ **Equivalence Ratio Maximum**; if ϕ is outside of this range, there is no soot formation.

Activation Temperature of Soot Formation Rate
is the term E/R in Equation 14–128 in the Theory Guide.

Magnussen Constant for Soot Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14–130 and Equation 14–131 in the Theory Guide).

Note that the default values for these parameters are for propane fuel [*15*] (p. 2209), [*98*] (p. 2214), and are considered to be valid for a wide range of hydrocarbon fuels.

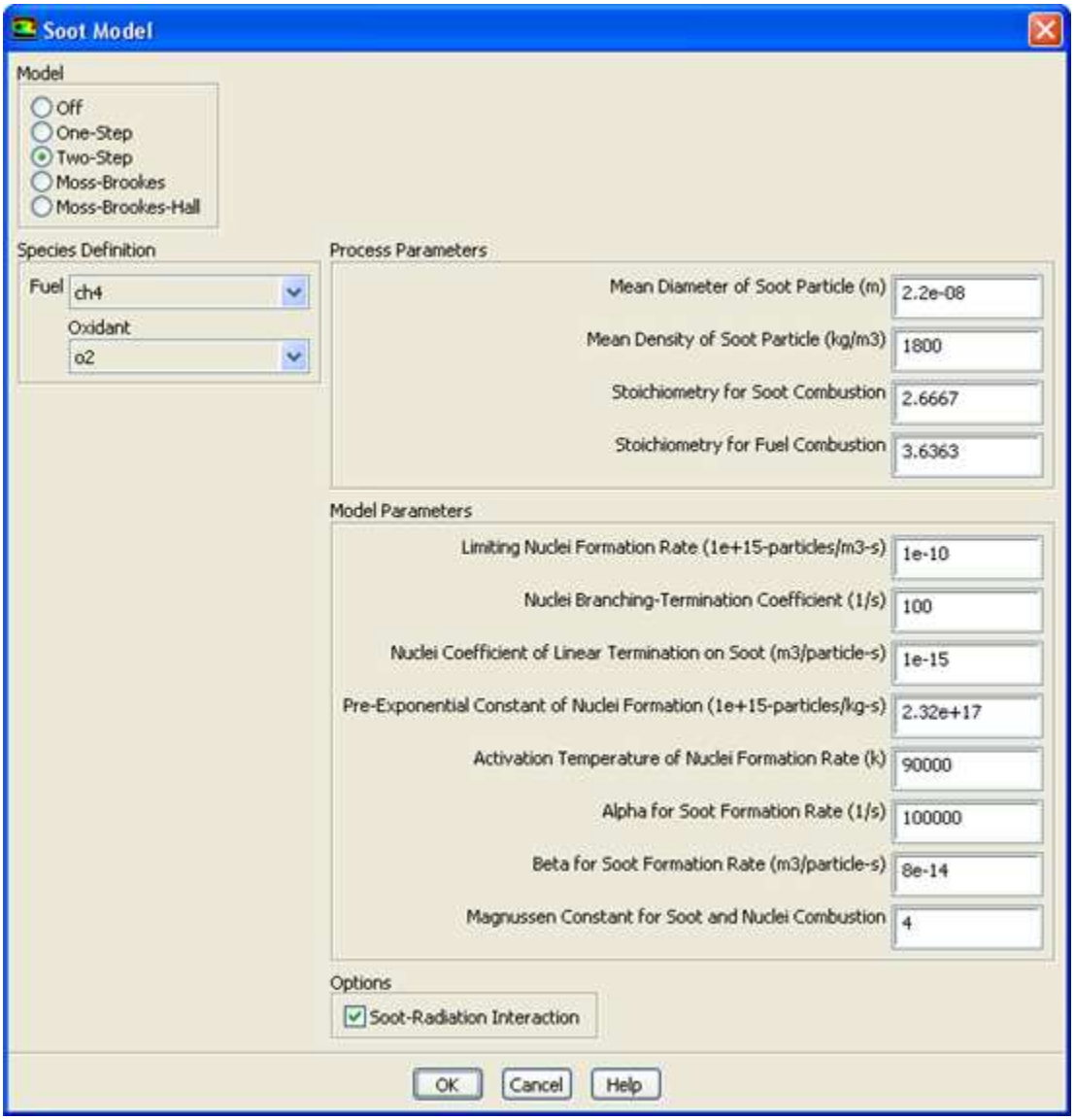
23.3.1.2. Setting Up the Two-Step Model

You can enable and set up the two-step soot formation model by using the *Soot Model Dialog Box* (p. 1717) (*Figure 23.13* (p. 976)).

Models → **Soot** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.13 The Soot Model Dialog Box for the Two-Step Model



Under **Model**, select **Two-Step**. The dialog box will expand to show the appropriate inputs.

Important

Note that the two-step Tesner model should only be used when the eddy-dissipation model is used to define the turbulence-chemistry interaction.

Next, you need to tell ANSYS FLUENT which chemical species in your model should be used as the fuel and oxidizer. Under **Species Definition**, select the fuel in the **Fuel** drop-down list and the oxidizer in the **Oxidant** drop-down list. If you are using the non-premixed model for the combustion calculation and your fuel stream consists of a mixture of components, you should choose the most appropriate species as the **Fuel** species for the soot formation model. Similarly, the most significant oxidizing component (e.g., O_2) should be selected as the **Oxidant**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

You must next define the **Process Parameters**, by inputting the stoichiometry of the fuel and soot combustion, as well as the average size and density of the soot particles, for the two-step model:

Mean Diameter of Soot Particle
and **Mean Density of Soot Particle** are the assumed average diameter and average density of the soot particles in the combustion system, used to compute the soot particle mass, m_p, in Equation 14–134 in the Theory Guide for the two-step model. Note that the default values for soot density and diameter are taken from [*49*] (p. 2211).

Stoichiometry for Soot Combustion
is the mass stoichiometry, v_{soot}, in Equation 14–131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS FLUENT (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion
is the mass stoichiometry, v_{fuel}, in Equation 14–131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS FLUENT (3.6363) is for combustion of propane (C_3H_8) by oxygen (O_2).

You must then set the **Modeling Parameters** that are used in Equation 14–130, Equation 14–131, Equation 14–134, Equation 14–136, and Equation 14–137 in the Theory Guide:

Limiting Nuclei Formation Rate
is the limiting value of the kinetic nuclei formation rate, η_0 in Equation 14–137 in the Theory Guide. Below this limiting value, the branching and termination term, $(f-g)$ in Equation 14–136 in the Theory Guide, is not included.

Nuclei Branching-Termination Coefficient
is the term $(f-g)$ in Equation 14–136 in the Theory Guide.

Nuclei Coefficient of Linear Termination on Soot
is the term g_0 in Equation 14–136 in the Theory Guide.

Pre-Exponential Constant of Nuclei Formation
is the pre-exponential term a_0 in the kinetic nuclei formation term, Equation 14–137 in the Theory Guide.

Activation Temperature of Nuclei Formation Rate
is the term E/R in the kinetic nuclei formation term, Equation 14–137 in the Theory Guide.

Alpha for Soot Formation Rate
is α, the constant in the soot formation rate equation, Equation 14–134 in the Theory Guide.

Beta for Soot Formation Rate
is β, the constant in the soot formation rate equation, Equation 14–134 in the Theory Guide.

Magnussen Constant for Soot and Nuclei Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14–130 and Equation 14–131 in the Theory Guide).

The default values for the two-step model are the same as in Magnussen and Hjertager [*49*] (p. 2211) (for an acetylene flame), except for a_0, which is assumed to have the original value from Tesner et al. [*95*] (p. 2214). If your model involves propane fuel rather that acetylene, it is recommended that you change the value of

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

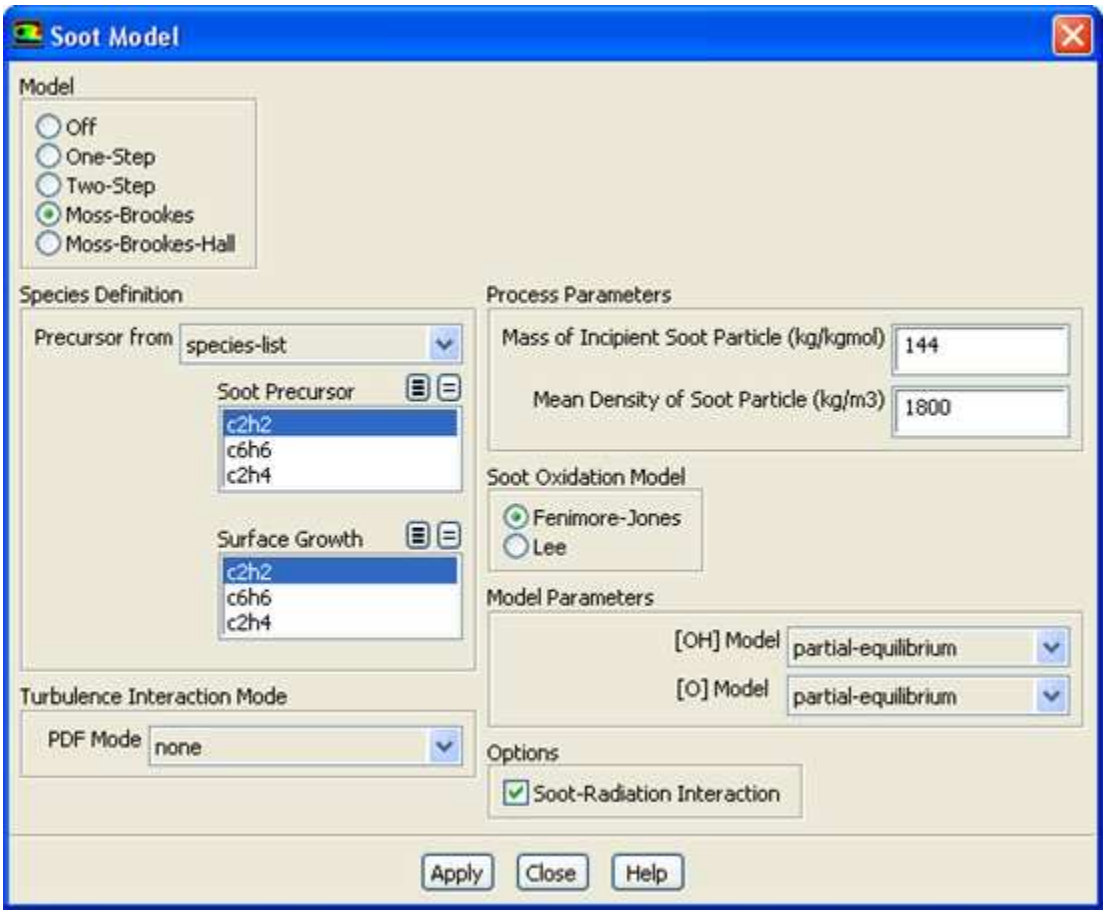
α to 3.5×10^8 [5] (p. 2209). For best results, you should modify both of these parameters, using empirically determined inputs for your specific combustion system.

23.3.1.3. Setting Up the Moss-Brookes Model and the Hall Extension

You can enable and set up the Moss-Brookes and Moss-Brookes-Hall soot formation models by using the *Soot Model Dialog Box* (p. 1717) (*Figure 23.14* (p. 978)).

Models → **Soot** → **Edit...**

Figure 23.14 The Soot Model Dialog Box for the Moss-Brookes Model



Under **Model**, select **Moss-Brookes** or **Moss-Brookes-Hall**. The dialog box will expand to show the appropriate inputs. Note the following about these models:

- The **Moss-Brookes** model was originally proposed for soot prediction in methane flames. However, it can be equally applicable to higher hydrocarbon species by appropriately modifying the soot precursor and participating surface growth species.
- The **Moss-Brookes-Hall** model is applicable for higher hydrocarbon fuels (e.g., kerosene) and will only be available when C_2H_2, C_6H_6, C_6H_5, and H_2 are present in the gas phase species list.

You must next define the precursor species in the **Species Definition** group box. When suitable precursor species are present in the species list, you can select **species-list** from the **Precursor from** drop-down list, and then select the **Soot Precursor** species and the **Surface Growth** species from the selection lists. Note that for the **Moss-Brookes** model, you can select acetylene (**c2h2**), ethylene (**c2h4**), and/or benzene (**c6h6**)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the **Soot Precursor**; if neither are present or if you would like to specify a different precursor correlation, then curve fitting will be used to determine the precursor and surface growth species mass fractions (see *Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation* (p. 981) for further details regarding curve fitting).

Next, specify how turbulent fluctuations will be accounted for in the soot formation calculations, by defining the turbulence parameters in the **Turbulence Interaction Mode** group box.

Select one of the options in the **PDF Mode** drop-down list:

- Select **none** to ignore turbulence and use laminar soot rate calculations.
- Select **temperature** to take into account fluctuations of temperature.
- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s). This is the recommended approach, as it generally yields the best results and accuracy.

Important

When modeling the formation of other pollutants along with soot, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** tab of the *NOx Model Dialog Box* (p. 1706) and the **Turbulence Interaction Mode** group boxes of the **SOx Model** and **Soot Model** dialog boxes. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous soot production rates are calculated at each cell. The PDF used for convoluting the instantaneous soot rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF in terms of mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous soot rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** group box:

PDF Type

allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 14–108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 14–111 in the Theory Guide.

PDF Points

allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 14–105 or Equation 14–106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (e.g., 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 14–114 of the Theory Guide). You have the option of selecting **transported** to instead solve Equation 14–113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option

provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature. The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field. You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**. You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**. Finally, if you have compiled a user-defined function for the soot rate and loaded the library into ANSYS FLUENT, then you can select **user-defined** so that the limit is specified by a UDF.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the soot formation calculations.

Important

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 14–114 in the Theory Guide).

Under **Process Parameters**, you must enter information about the mass and mean density of the soot particles:

Mass of Incipient Soot Particles

is M_P in Equation 14–142 and Equation 14–144 in the Theory Guide. Note that this value was assumed to be 144 kg/kmol (12 carbon atoms) in the work of Brookes and Moss, whereas the Hall extension model assumed it to be 1200 kg/kmol (100 carbon atoms).

Mean Density of Soot Particle

is ρ_{soot} in Equation 14–141 in the Theory Guide and ρ in Equation 14–144 in the Theory Guide. Note that this value was assumed to be 1800 kg/m^3 in the work of Brookes and Moss [*12*] (p. 2209), whereas Hall et al. [*30*] (p. 2210) assumed it to be 2000 kg/m^3.

Next, you must select the **Soot Oxidation Model**. Your choices include the **Fenimore-Jones** model, as originally used in Brookes and Moss' work, or the **Lee** extended model. The **Lee** model will model soot oxidation due to hydroxyl radicals as in the **Fenimore-Jones** model, as well as the oxidation due to molecular oxygen.

You must then set the **Modeling Parameters**:

[OH] Model

allows you to specify the method by which the OH radical concentration is calculated. The recommended selection from the drop-down list is **instantaneous**, although this option is only available when OH is available in the species list and is calculated by the combustion model. The other option is the **partial-equilibrium** model, which necessitates the availability of $[\mathrm{O}]$ atom concentration within the field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[O] Model

must be defined when you have selected **partial-equilibrium** for the **[OH] Model**, and specifies the method by which the O radical concentration is calculated. The options include **equilibrium**, **partial-equilibrium**,and **instantaneous**.

Note that in ANSYS FLUENT, the oxidation rate scaling parameter (C_{oxid} in Equation 14–142 in the Theory Guide) is set to unity. If you would like to change the value of this parameter, you can use the `define/models/soot-parameters` text command. A lower value will reduce the amount of soot oxidation.

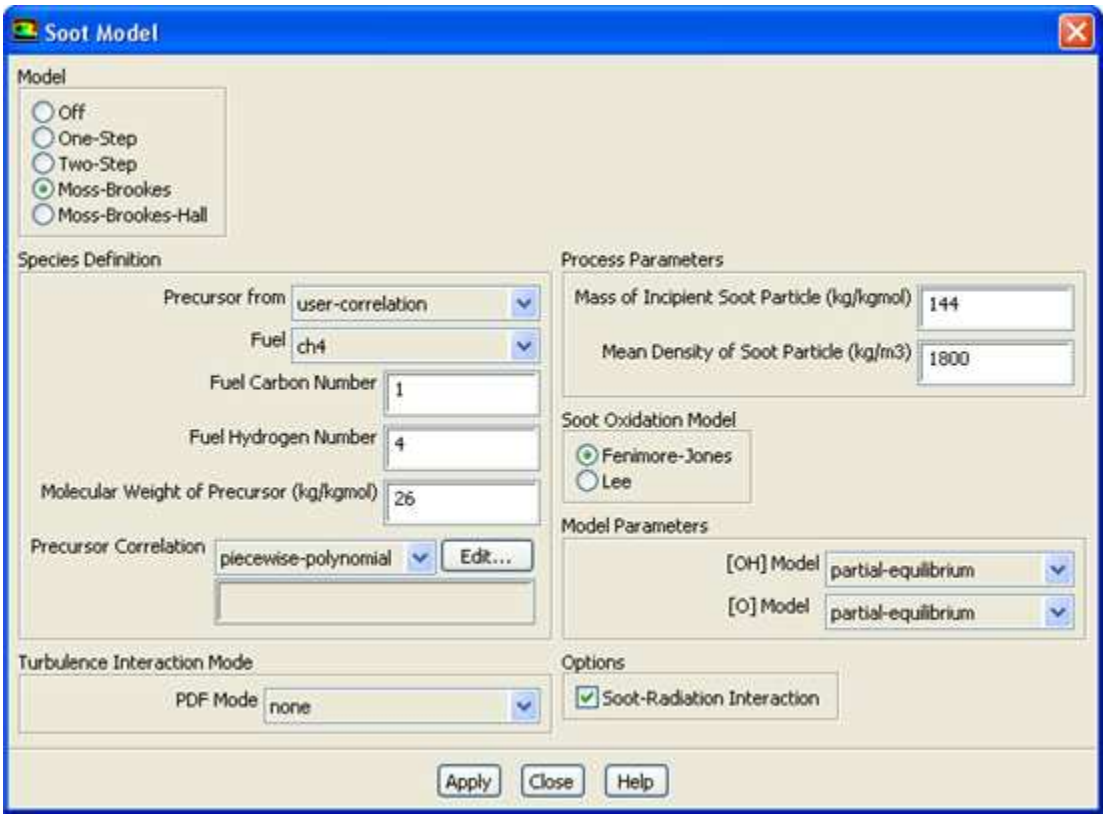
If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

23.3.1.3.1. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation

ANSYS FLUENT accepts the following as possible precursor species for the Moss-Brookes model: $\mathrm{C_2H_2}$, $\mathrm{C_6H_6}$, and $\mathrm{C_2H_4}$. If none of these species are present in the species list (as is often the case when using the eddy-dissipation model) or if you would prefer to specify a different precursor correlation, your setup for the Moss-Brookes model will be different than noted previously. Under such circumstances, you should select **user-correlation** from the **Precursor from** drop-down list in the **Species Definition** group box (note that this is only option possible when the appropriate species are not present). The *Soot Model Dialog Box* (p. 1717) will then be as shown in *Figure 23.15* (p. 982). The parameters you set in the **Species Definition** group box allow ANSYS FLUENT to calculate a mixture fraction based on the mass fractions of the oxidant and the carbon/hydrogen contributed by a designated fuel species. The precursor species mass fraction will then be computed as a function (which you will also define) of this mixture fraction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.15 The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation



In the **Species Definition** group box, you will first select a **Fuel** species and enter the related **Fuel Carbon Number** and **Fuel Hydrogen Number** for use in the mixture fraction calculation. Next, enter the **Molecular Weight of Precursor** (the default value is for acetylene). Then make a selection in the **Precursor Correlation** drop-down list to indicate how the precursor mass fraction will be related to the mixture fraction. A **piecewise-polynomial** profile is defined by default.

Important

Note that the default values for the **piecewise-polynomial** profile are only valid for a methane diffusion flame simulation, in which both the air and fuel initial temperatures are set to 290 K, and acetylene is assumed as the soot precursor.

If you decide not to use the default values for **Precursor Correlation**, you must define the correlation between the precursor mass fraction and the mixture fraction. This correlation should be based on a laminar flamelet profile that you have generated: if possible, you should generate the profile using the **Species Model** dialog box (as described in *Setting Up the Steady and Unsteady Laminar Flamelet Models* (p. 847)), being sure to select **Steady Flamelet** in the **Chemistry** tab; if there is no mechanism, you can generate the profile using the equilibrium chemistry model (see *Setting Up the Equilibrium Chemistry Model* (p. 843) for details); you may also use a third-party software package of your choosing. You should then apply a curve-fitting technique to your generated profile, to obtain either a constant value or a piecewise-polynomial function.

In a piecewise-polynomial function, the laminar flamelet profile is divided into a number of mixture fraction ranges. In each range, the precursor species mass fraction Y_{prec} is defined using the following equation:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

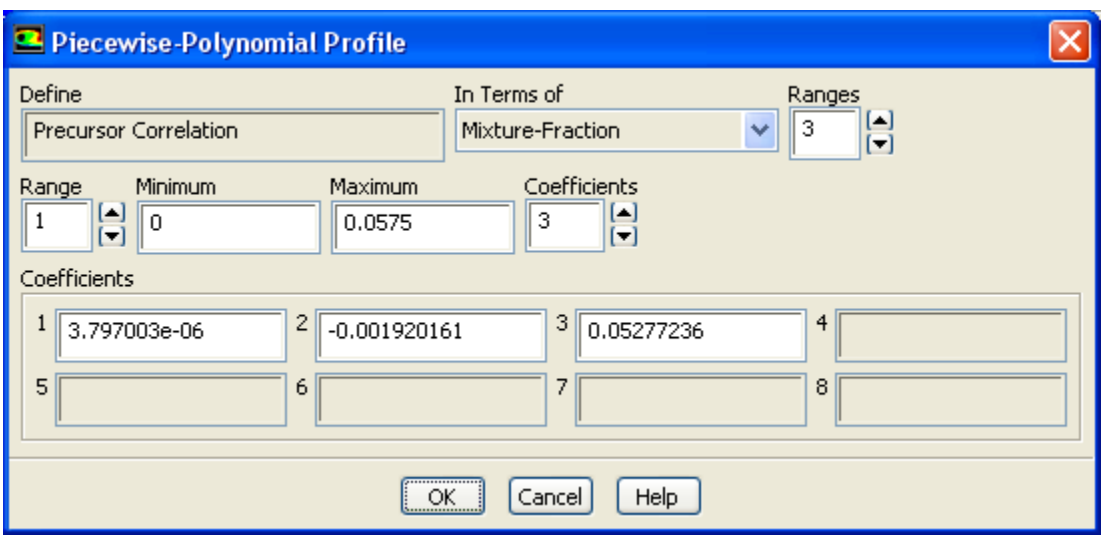
$$Y_{prec} = \sum_{i=1}^{i=NC} C_i f^{i-1} \tag{23–14}$$

where NC is the number of coefficients C, and f is the mixture fraction. The following piecewise-polynomial function corresponds to the default settings in ANSYS FLUENT:

$$Y_{prec} = \begin{cases} \text{for } 0 \le f < 0.0575: \\ 3.797003 \times 10^{-6} - 1.920161 \times 10^{-3} f \\ +5.277237 \times 10^{-2} f^2 \\ \text{for } 0.0575 \le f < 0.128: \\ 1.051312 - 71.34743 f + 1964.038 f^2 + 281825.9 f^3 \\ +223543.4 f^4 + 932192 f^5 + 1599627 f^6 \\ \text{for } 0.128 \le f < 1: \\ 7.988928 \times 10^{-3} - 8.440912 \times 10^{-3} f \\ +4.273195 \times 10^{-4} f^2 \end{cases} \tag{23–15}$$

To define a piecewise-polynomial profile to relate the precursor mass fraction to the mixture fraction, select **piecewise-polynomial** from the **Precursor Correlation** drop-down list and click the **Edit...** button. The *Piecewise-Polynomial Profile Dialog Box* (p. 1752) (*Figure 23.16* (p. 983)) will open.

Figure 23.16 The Piecewise-Polynomial Profile Dialog Box



Then perform the following steps in the **Piecewise-Polynomial Profile** dialog box:

1. Enter the number of **Ranges**. For the example shown in *Equation 23–15* (p. 983), three ranges of mixture fraction are defined, which is the maximum allowed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For the first range (**Range** = 1), enter the **Minimum** and **Maximum** mixture fraction values, and the number of **Coefficients**. (Up to eight coefficients are available.) The number of coefficients defines the order of the polynomial. An input of 1 defines a polynomial of order 0, and the mass fraction will be constant and equal to the single coefficient. An input of 2 defines a polynomial of order 1, and the mass fraction will vary linearly with mixture fraction, and so on.
3. Define the values for the coefficients in the **Coefficients** group box. The dialog box in *Figure 23.16* (p. 983) shows the inputs for the first range of *Equation 23–15* (p. 983).
4. Increase the value of **Range** and enter the **Minimum** and **Maximum** mixture fractions, number of **Coefficients**, and the values for the **Coefficients** for the next range. Repeat if there is a third range.

Important

Note when defining the ranges, you must start with the lowest mixture fraction range, and then proceed in order to the highest range. The solver will *not* sort them for you.

To define a constant profile to relate the precursor mass fraction to the mixture fraction, select **constant** from the **Precursor Correlation** drop-down list and enter a value in the accompanying text entry box.

23.3.1.4. Defining Boundary Conditions for the Soot Model

At flow inlet boundaries, you will need to specify the **Soot Mass Fraction** and (when not using the one-step model) the **Nuclei** mass concentration. These correspond to Y_{soot} in Equation 14–126 and Equation 14–140 and b^*_{nuc} in Equation 14–132 and Equation 14–140 in the Theory Guide, respectively.

Boundary Conditions

You can retain the default inlet values of zero for both quantities or you can input nonzero numbers as appropriate for your combustion system.

23.3.1.5. Reporting Soot Quantities

ANSYS FLUENT provides additional reporting options when your model includes soot formation. You can generate graphical plots or alphanumeric reports of the following items:

- **Mass fraction of Soot**
- **Mole fraction of Soot**
- **Soot Density**
- **Soot Volume fraction**
- **Rate of Soot**
- **Normalized Concentration of Nuclei** (unavailable for one-step model)
- **Rate of Nuclei** (unavailable for one-step model)
- **Rate of Soot Mass Nucleation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Surface Growth** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Oxidation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Nucleation** (Moss-Brookes and Moss-Brookes-Hall models only)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rate of Coagulation** (Moss-Brookes and Moss-Brookes-Hall models only)

These parameters are contained in the **Soot...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

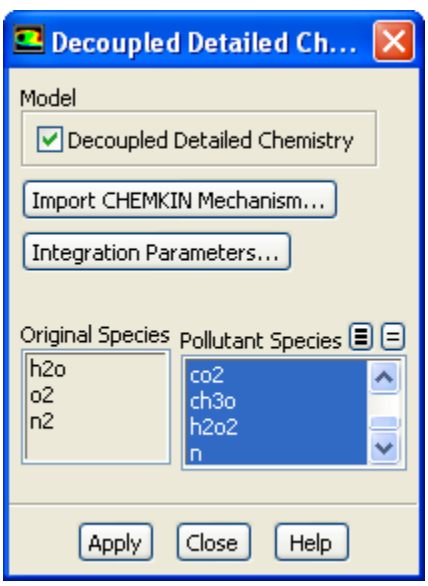
23.4. Using the Decoupled Detailed Chemistry Model

The decoupled detailed chemistry pollutant model is used to postprocess slowly-forming, trace pollutant species on a steady-state flow field using detailed chemical kinetic mechanisms. For theoretical information, please refer to Decoupled Detailed Chemistry Model. The recommended procedure is as follows:

1. Calculate your steady combustion problem using the Species Transport, Non-Premixed, Partially-Premixed, or PDF Transport models in ANSYS FLUENT. It is recommended that you save your case and data file, as it may be difficult to revert to the original settings after you set up the decoupled detailed chemistry pollutant model.
2. Enable the **Decoupled Detailed Chemistry** option in the **Decoupled Detailed Chemistry** dialog box. To access this dialog box, make sure that either the non-premixed or partially premixed model is selected, or that reactions are enabled if the species transport or PDF transport models are used.

   Models → **Decoupled Detailed Chemistry** → **Edit...**

   Figure 23.17 The Decoupled Detailed Chemistry Dialog Box



3. Click **Import CHEMKIN Mechanism...** to import your detailed chemistry mechanism in CHEMKIN format. The **CHEMKIN Mechanism Import** dialog box (described in *Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827)) will open, where you will browse to the file. After the CHEMKIN mechanism is imported, the **Decoupled Detailed Chemistry** dialog box will expand to include the species.
4. Select the **Pollutant Species** and click **Apply**. The pollutant species are typically slowly forming (far from chemical equilibrium), and occur at miniscule mass fractions. ANSYS FLUENT will create a mixture material called **pollutant-mixture** and enable the **Species Transport** model with the **Stiff Chemistry Solver**. All species in the imported mechanism that were not in the original combustion model and are not pollutant species will be calculated by chemical equilibrium at the cell temperature. The solution of all transport equations, except the selected pollutant species, are disabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

Species which are not identified as pollutants and do not participate in the reactions amongst the pollutant species are eliminated. Similarly, reactions in the imported CHEMKIN mechanism, which do not include any pollutant species are also eliminated.

5. Click **Integration Parameters...** to open the **Integration Parameters** dialog box. Set the **ISAT Parameters**, such as the **ISAT Error Tolerance** and **Max. Storage**. An **ISAT Error Tolerance** of 1e-5 is recommended, and **Max. Storage** should be set to a large fraction of the free RAM memory available. To learn more about setting integration parameters, please refer to *Using ISAT* (p. 924).
6. Define the boundary conditions for all pollutants at flow inlets (these are usually zero).

Boundary Conditions

7. Iterate until convergence.

Run Calculation

8. Review the mass fractions of pollutants by generating graphical plots or alphanumeric reports in the usual way.
9. Save a new set of case and data files, if desired.

File → **Write** → **Case & Data...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 24: Predicting Aerodynamically Generated Noise

This chapter provides an overview of ANSYS FLUENT's approaches to computing aerodynamically generated sound, the model setup, and the procedure for computing sound. For more information about the underlying theory behind aerodynamically generated sound, see "Aerodynamically Generated Noise" in the Theory Guide.

24.1. Overview
24.2. Using the Ffowcs Williams and Hawkings Acoustics Model
24.3. Using the Broadband Noise Source Models

24.1. Overview

Considering the breadth of the discipline and the challenges encountered in aerodynamically generated noise, it is not surprising that a number of computational approaches have been proposed over the years whose sophistication, applicability, and cost widely vary.

ANSYS FLUENT offers three approaches to computing aerodynamically generated noise; a direct method, an integral method based on acoustic analogy and a method that utilizes broadband noise source models.

For additional information, please see the following sections:

24.1.1. Direct Method
24.1.2. Integral Method Based on Acoustic Analogy
24.1.3. Broadband Noise Source Models

24.1.1. Direct Method

In this method, both generation and propagation of sound waves are directly computed by solving the appropriate fluid dynamics equations. Prediction of sound waves always requires time-accurate solutions to the governing equations. Furthermore, in most practical applications of the direct method, one has to employ governing equations that are capable of modeling viscous and turbulence effects, such as unsteady Navier-Stokes equations (i.e., DNS), RANS equations, and filtered equations used in DES and LES.

The direct method is thus computationally difficult and expensive inasmuch as it requires highly accurate numerics, very fine computational meshes all the way to receivers, and acoustically nonreflecting boundary conditions. The computational cost becomes prohibitive when sound is to be predicted in the far field (e.g., hundreds of chord-lengths in the case of an airfoil). The direct method becomes feasible when receivers are in the near field (e.g., cabin noise). In many such situations involving near-field sound, sounds (or pseudo-sounds for that matter) are predominantly due to local hydrodynamic pressure which can be predicted with a reasonable cost and accuracy.

Since sound propagation is directly resolved in this method, one normally needs to solve the compressible form of the governing equations (e.g., compressible RANS equations, compressible form of filtered equations for LES). Only in situations where the flow is low subsonic and the receivers in the near field sense primarily local hydrodynamic pressure fluctuations (i.e., pseudo sound) can incompressible flow formulations be used. But this incompressible treatment will also not allow to simulate resonance and feedback phenomena.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.1.2. Integral Method Based on Acoustic Analogy

For predictions of mid- to far-field noise, the methods based on Lighthill's acoustic analogy [*44*] (p. 2211) offer viable alternatives to the direct method. In this approach, the near-field flow obtained from appropriate governing equations such as unsteady RANS equations, DES, or LES are used to predict the sound with the aid of analytically derived integral solutions to wave equations. The acoustic analogy essentially decouples the propagation of sound from its generation, allowing one to separate the flow solution process from the acoustics analysis.

ANSYS FLUENT offers a method based on the Ffowcs Williams and Hawkings (FW-H) formulation [*23*] (p. 2210). The FW-H formulation adopts the most general form of Lighthill's acoustic analogy, and is capable of predicting sound generated by equivalent acoustic sources such as monopoles, dipoles, and quadrupoles. ANSYS FLUENT adopts a time-domain integral formulation wherein time histories of sound pressure, or acoustic signals, at prescribed receiver locations are directly computed by evaluating a few surface integrals.

Time-accurate solutions of the flow-field variables, such as pressure, velocity components, and density on source (emission) surfaces, are required to evaluate the surface integrals. Time-accurate solutions can be obtained from unsteady Reynolds-averaged Navier-Stokes (URANS) equations, large eddy simulation (LES), or detached eddy simulation (DES) as appropriate for the flow at hand and the features that you want to capture (e.g., vortex shedding). The source surfaces can be placed not only on impermeable walls, but also on interior (permeable) surfaces, which enables you to account for the contributions from the quadrupoles enclosed by the source surfaces. Both broadband and tonal noise can be predicted depending on the nature of the flow (noise source) being considered, turbulence model employed, and the time scale of the flow resolved in the flow calculation.

The FW-H acoustics model in ANSYS FLUENT allows you to select multiple source surfaces and receivers. It also permits you either to save the source data for a future use, or to carry out an "on the fly" acoustic calculation simultaneously as the transient flow calculation proceeds, or both. Sound pressure signals thus obtained can be processed using the fast Fourier transform (FFT) and associated postprocessing capabilities to compute and plot such acoustic quantities as the overall sound pressure level (SPL) and power spectra.

One important limitation of ANSYS FLUENT's FW-H model is that it is applicable only to predicting the propagation of sound toward *free space*. Thus, while the model can be legitimately used to predict far-field noise due to external aerodynamic flows, such as the flows around ground vehicles and aircrafts, it cannot be used for predicting the noise propagation inside ducts or wall-enclosed space.

24.1.3. Broadband Noise Source Models

In many practical applications involving turbulent flows, noise does not have any distinct tones, and the sound energy is continuously distributed over a broad range of frequencies. In those situations involving *broadband noise*, statistical turbulence quantities readily computable from RANS equations can be utilized, in conjunction with semi-empirical correlations and Lighthill's acoustic analogy, to shed some light on the source of broadband noise.

ANSYS FLUENT offers several such *source models* that enable you to quantify the local contribution (per unit surface area or volume) to the total acoustic power generated by the flow. They include the following:

- Proudman's formula
- jet noise source model
- boundary layer noise source model
- source terms in the linearized Euler equations
- source terms in Lilley's equation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Considering that one would ultimately want to come up with some measures to mitigate the noise generated by the flow in question, the source models can be employed to extract useful diagnostics on the noise source to determine which portion of the flow is primarily responsible for the noise generation. Note, however, that these source models do not predict the sound at receivers.

Unlike the direct method and the FW-H integral method, the broadband noise source models do not require transient solutions to any governing fluid dynamics equations. All the source models need is what typical RANS models would provide, such as the mean velocity field, turbulent kinetic energy (k) and the dissipation rate (ε). Therefore, the use of broadband noise source models requires the least computational resources.

24.2. Using the Ffowcs Williams and Hawkings Acoustics Model

The procedure for computing sound using the FW-H acoustics model in ANSYS FLUENT consists largely of two steps. In the first step, a time-accurate flow solution is generated from which time histories of the relevant variables (e.g., pressure, velocity, and density) on the selected source surfaces are obtained. In the second step, sound pressure signals at the user-specified receiver locations are computed using the source data collected during the first step.

Important

Note that you can also use the FW-H model for a steady-state simulation in the case where your model has a single moving reference frame. Here, the thickness and loading noise due to the motion of the noise sources is computed using the FW-H integrals (see Equation 15–5 and Equation 15–6 in the Theory Guide), except that the term involving the time derivative of surface pressure (contribution to $\dot{L}_r$ in Equation 15–6 in the Theory Guide) is set to zero.

In computing sound pressure using the FW-H integral solution, ANSYS FLUENT uses a so-called "forward-time projection" to account for the time delay between the emission time (the time at which the sound is emitted from the source) and the reception time (the time at which the sound arrives at the receiver location). The forward-time projection approach enables you to compute sound at the same time "on the fly" as the transient flow solution progresses, without having to save the source data.

In this section, the procedure for setting up and using the FW-H acoustics model is outlined first, followed by detailed descriptions of each of the steps involved. Remember that only the steps that are pertinent to acoustics modeling are discussed here. For information about the inputs related to other models that you are using in conjunction with the FW-H acoustics model, see the appropriate sections for those models.

The general procedure for carrying out an FW-H acoustics calculation in ANSYS FLUENT is as follows:

1. Calculate a converged flow solution. For a transient case, run the transient solution until you obtain a "statistically steady-state" solution as described below.
2. Enable the FW-H acoustics model and set the associated model parameters.

   Models → **Acoustics** → **Edit...**
3. Specify the source surface(s) and choose the options associated with acquisition and saving of the source data. For a steady-state case, specify the rotating surface zone(s) as the source surface(s).
4. Specify the receiver location(s).
5. Continue the transient solution for a sufficiently long period of time and save the source data (transient cases only).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Calculation

6. Compute and save the sound pressure signals.

   Run Calculation → **Acoustic Signals...**

7. Postprocess the sound pressure signals.

   Plots → **FFT** → **Set Up...**

Important

Before you start the acoustics calculation for a transient case, an ANSYS FLUENT transient solution should have been run to a point where the transient flow field has become "statistically steady". In practice, this means that the unsteady flow field under consideration, including all the major flow variables, has become fully developed in such a way that its statistics do not change with time. Monitoring the major flow variables at selected points in the domain is helpful for determining if this condition has been met.

As discussed earlier, URANS, DES, and LES are all legitimate candidates for transient flow calculations. For stationary source surfaces, the frequency of the aerodynamically generated sound heard at the receivers is largely determined by the time scale or frequency of the underlying flow. Therefore, one way to determine the time-step size for the transient computation is to make it small enough to resolve the smallest characteristic time scale of the flow at hand that can be reproduced by the mesh and turbulence adopted in your model.

Once you have obtained a statistically stationary flow-field solution, you are ready to acquire the source data.

For additional information, please see the following sections:

24.2.1. Enabling the FW-H Acoustics Model
24.2.2. Specifying Source Surfaces
24.2.3. Specifying Acoustic Receivers
24.2.4. Specifying the Time Step
24.2.5. Postprocessing the FW-H Acoustics Model Data

24.2.1. Enabling the FW-H Acoustics Model

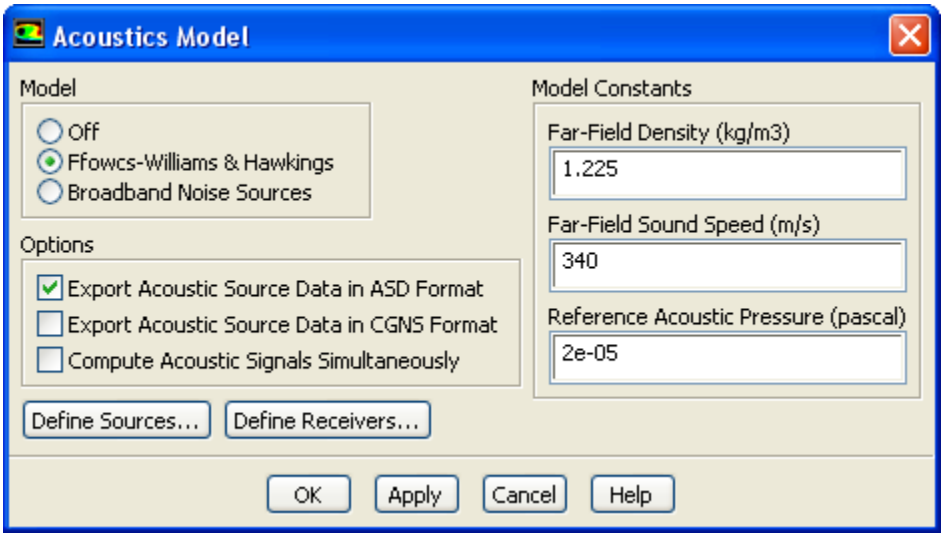
To enable the FW-H acoustics model, select **Ffowcs-Williams & Hawkings** in the **Acoustics Model** dialog box (*Figure 24.1* (p. 991)).

Models → **Acoustics** → **Edit...**

When you select **Ffowcs-Williams & Hawkings**, the dialog box will expand to show the relevant fields for user inputs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.1 The Acoustics Model Dialog Box



24.2.1.1. Setting Model Constants

Under **Model Constants** in the **Acoustics Model** dialog box, specify the relevant acoustic parameters and constants used by the model.

Far-Field Density

(for example, ρ_0 in Equation 15–1 in the Theory Guide) is the far-field fluid density.

Far-Field Sound Speed

(for example, a_0 in Equation 15–1) is the sound speed in the far field (= $\sqrt{\gamma R T_0}$).

Reference Acoustic Pressure

(for example, p_{ref} in *Equation 31–39* (p. 1506)) is used to calculate the sound pressure level in dB (see *Using the FFT Utility* (p. 1503)). The default reference acoustic pressure is 2×10^{-5} Pa.

Number of Time Steps Per Revolution

is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.

Number of Revolutions

is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of revolutions that will be simulated in the model.

Source Correlation Length

is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depth-wise direction using the identical source data (see *Figure 24.2* (p. 993)).

The default values are appropriate for sound propagating in air at atmospheric pressure and temperature.

24.2.1.2. Computing Sound "on the Fly"

The FW-H acoustics model in ANSYS FLUENT allows you to perform simultaneous calculation of the sound pressure signals at the prescribed receivers without having to write the source data to files, which can save a significant amount of disk space on your machine. To enable this "on-the-fly" calculation of sound, enable the **Compute Acoustic Signals Simultaneously** option in the **Acoustics Model** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Because the noise computation takes a negligible percentage of memory and computational time compared to a transient flow calculation, this option can be used by itself or along with the process of source data file export and sound calculation. For the latter, computing signals "on the fly" allows you to see when the signals have become statistically steady so you can know when to stop the simulation.

When the **Compute Acoustic Signals Simultaneously** option is enabled, the ANSYS FLUENT console window will print a message at the end of each time step indicating that the sound pressure signals have been computed (e.g., `Computing sound signals at x receiver locations...`, where `x` is the number of receivers you specified). Enabling this option instructs ANSYS FLUENT to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Important

Note that this option is available only when the FW-H acoustics model has been enabled. See below for details about exporting source data without enabling the FW-H model.

24.2.1.3. Writing Source Data Files

Although the "on-the-fly" capability is a convenient feature, you will want to save the source data as well, because the acquisition of source data during a transient flow-field calculation is the most time-consuming part of acoustics computations, and you most likely will not want to discard it. By saving the source data, you can always reuse it to compute the sound pressure signals at new or additional receiver locations.

To save the source data to files, enable either the **Export Acoustic Source Data in ASD Format** or the **Export Acoustic Source Data in CGNS Format** option, or both. After you have made your selection, the relevant source data at all face elements of the selected source surfaces will be written into the files you specify. The source data vary depending on the solver option you have chosen and whether the source surface is a wall or not. *Table 24.1: Source Data Saved in Source Data Files* (p. 993) shows the flow variables saved as the source data.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.2 The Acoustics Model Dialog Box for a 2D Steady-State Case with a Single Moving Reference Frame

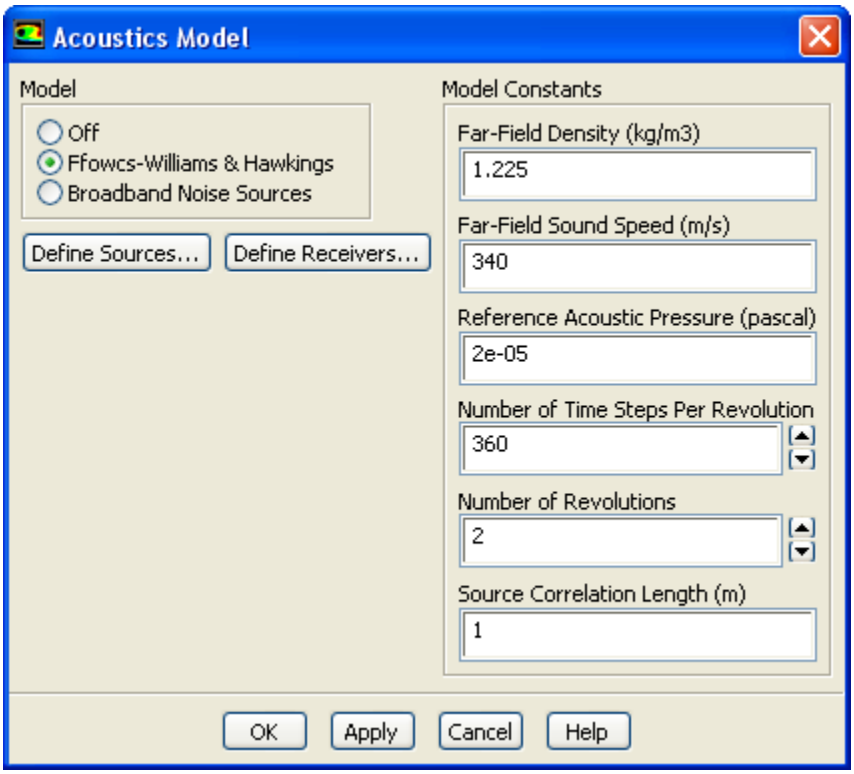


Table 24.1 Source Data Saved in Source Data Files

Solver Option	Source Surface	Source Data
incompressible	walls	p
incompressible	permeable surfaces	p, u, v, w
compressible	walls	p
compressible	permeable surfaces	ρ, p, u, v, w

See *Specifying Source Surfaces* (p. 995) for details on how to specify parameters for exporting source data.

24.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS FLUENT ASD Format

You can export sound source data for use with SYSNOISE without having to enable the Ffowcs Williams and Hawkings (FW-H) model. You will still need to specify source surfaces (see *Specifying Source Surfaces* (p. 995)), as `.index` and `.asd` files are required by SYSNOISE. In addition, you can choose fluid zones as emission sources if you want to export quadrupole sources. To enable the selection of fluid zones as sources, use the

`define` → `models` → `acoustics` → `export-volumetric-sources?`

text command and change the selection to `yes`.

SYSNOISE also requires centroid data for source zones that are being exported.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For fan noise calculations, once you have specified the source zones in the **Acoustic Sources** dialog box and you have selected **Export Acoustic Source Data in ASD Format** from the **Acoustics Model** dialog box, you can export geometry in cylindrical coordinates by using the

`define` → `models` → `acoustics` → `cylindrical-export?`

text command and changing the selection to `yes`. By default, ANSYS FLUENT exports source zones for SYSNOISE in Cartesian coordinates.

You can then export the centroid data to a data file using the following text command:

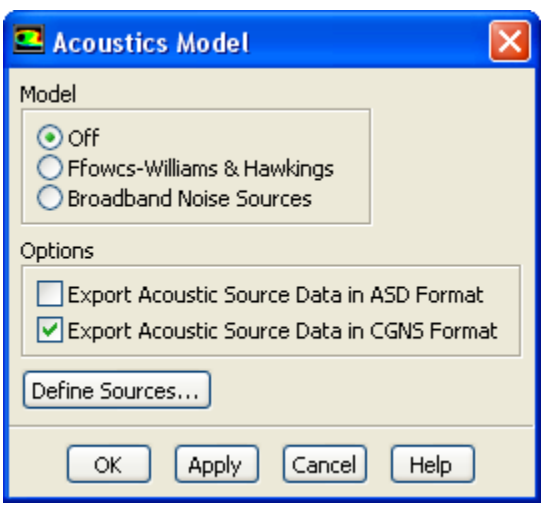
`define` → `models` → `acoustics` → `write-centroid-info`

Since you will not be using the FW-H model to compute signals, you will not need to specify any acoustic model parameters or receiver locations. Also, you will not be able to enable the **Compute Acoustic Signals Simultaneously** option in the **Acoustics Model** dialog box, and **Acoustic Signals...** will not be available in the **Run Calculation** task page.

24.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format

The sound source data for non-permeable surfaces can be exported in the CGNS file format (for Virtual Lab) without having to enable the Ffowcs Williams and Hawkings (FW-H) model. Enable the **Export Acoustic Source Data in CGNS Format** option in the **Acoustics Model** dialog box (*Figure 24.3* (p. 994)). Specify the source surfaces in the **Acoustics Sources** dialog box (see *Specifying Source Surfaces* (p. 995)) where, by default, the **Number of Time Steps per File** is set to 1.

Figure 24.3 The Acoustics Model Dialog Box for Exporting in CGNS Format



Virtual Lab requires a mesh data file (named `<prefix>_mesh.cgns`) and a solution data file (named `<prefix>_<timestep>.cgns`). The string `<prefix>` is a generic name, which you will specify in the **Source Data Root File Name** in the **Acoustics Sources** dialog box. There is one single solution data file (`.cgns`) per time level exported, which contains the static pressure at the wall-face centroid location. The `.cgns` files will be stored in a directory, which you specify (named `<directory_name>/<prefix>`) in the **Source Data Root File Name**.

In addition, you can export quadrupole sources data by choosing fluid zones as emission sources. To enable the selection of fluid zones as sources, use the text command:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

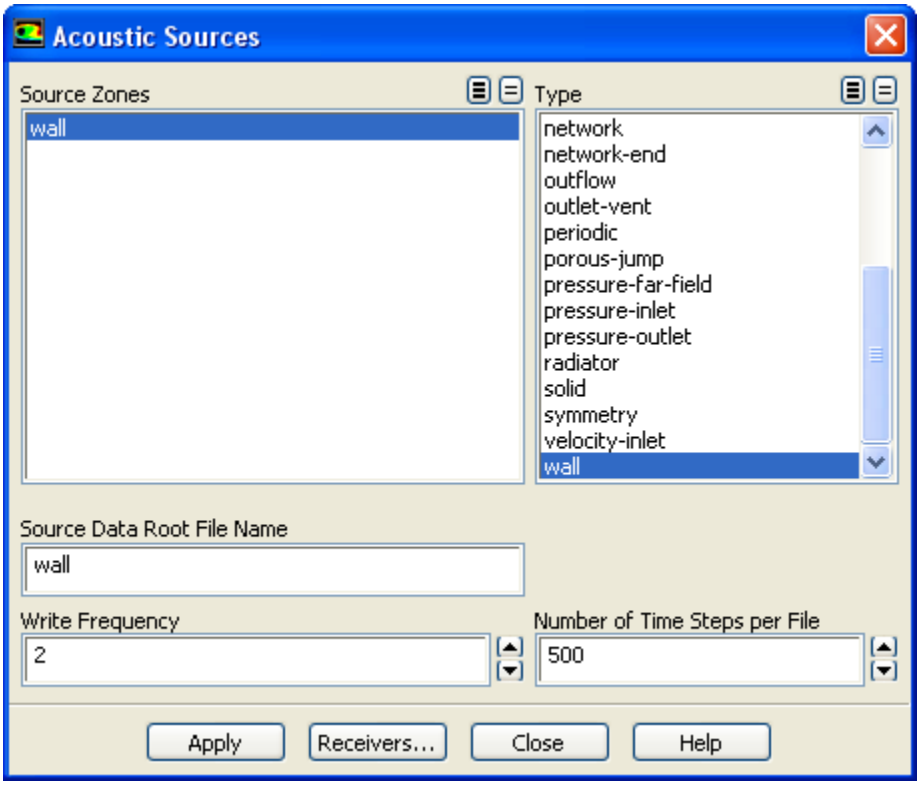
`define` → `models` → `acoustics` → `export-volumetric-sources-cgns?`

When asked if you would like to `Export volumetric sources?` enter `yes`. Note that Virtual Lab requires volumetric mesh data file (`<prefix>_Q_mesh.cgns`) and quadrupole solution data files (`<prefix>_Q_<timestep>.cgns`). The `.cgns` file will be stored in a similar way to that of dipole data export, in the directory specified by you in the **Source Data Root File Name** text entry box.

24.2.2. Specifying Source Surfaces

In the **Acoustics Model** dialog box, click the **Define Sources...** button to open the **Acoustic Sources** dialog box (*Figure 24.4* (p. 995)). Here you will specify the source surface(s) to be used in the acoustics calculation and the inputs associated with saving source data to files.

Figure 24.4 The Acoustic Sources Dialog Box



Under **Source Zones**, you can select multiple emission (source) surfaces and the surface **Type** that you can select is *not* limited to a **wall**. You can also choose **interior** surfaces and sliding interfaces (both stationary and rotating) as source surfaces.

Important

The ability to choose multiple source surfaces is useful for investigating the contributions from individual source surfaces. The results based on the use of multiple source surfaces are valid as long as there are negligible acoustic interactions among the surfaces. Thus, some caution needs to be taken when selecting multiple source surfaces.

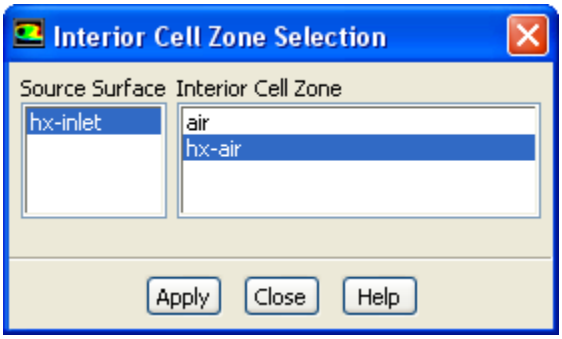
In cases where multiple source surfaces are selected, no source surface may enclose any of the other source surfaces. Otherwise, the sound pressure calculated based on the source surfaces will not be accurate, as the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

contribution from the enclosed (inner) source surfaces is over predicted, since the FW-H model is unable to account for the shading of the sound from the inner source surfaces by the enclosure surface.

If you specify any interior surfaces as source surfaces, the interior surface must be generated in advance (e.g., in GAMBIT) in such a way that the two cell zones adjacent to the surface have different cell zone IDs. Furthermore, you must correctly specify which of the two zones is occupied by the quadrupole sources (interior cell zone). This will allow ANSYS FLUENT to determine the direction in which the sound will propagate. When you first attempt to select a legitimate interior surface (i.e., an interior surface having two different cell zones on both sides) as a source surface, the *Interior Cell Zone Selection Dialog Box* (p. 1734) (*Figure 24.5* (p. 996)) will appear. You will then need to select the interior cell zones from the two zones listed under the **Interior Cell Zone**. *Figure 24.6* (p. 997) shows an example of an interior source surface.

Figure 24.5 The Interior Cell Zone Selection Dialog Box



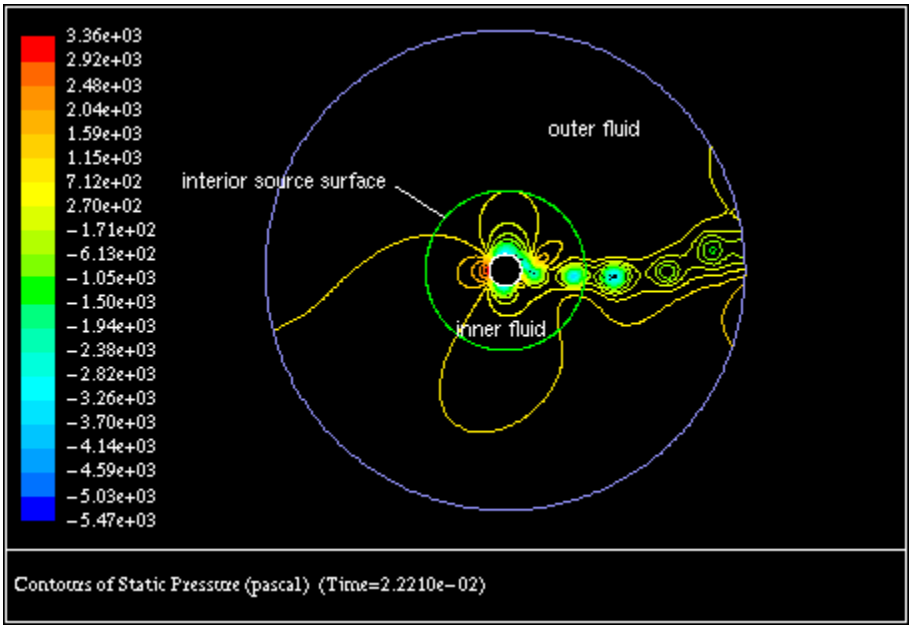
Like general interior surfaces, if the source surfaces selected are sliding interfaces, a dialog box similar to *Figure 24.5* (p. 996) will appear that will show the two adjacent cell zones and you will be asked to specify the zone which has the sound sources.

Important

When a permeable surface (either interior or sliding interface) is chosen as the source surface, other wall surfaces inside the volume enclosed by the permeable surface that generate sound should not be chosen for the acoustics calculation. For example, when running an "on-the-fly" calculation, if both these surfaces are selected, the sound pressure will be counted twice.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.6 An Interior Source Surface



24.2.2.1. Saving Source Data

To save the source data, you have to specify the **Source Data Root File Name**, **Write Frequency** (in number of time steps), and **Number of Time Steps per File** in the **Acoustic Sources** dialog box.

The **Source Data Root File Name** is used to give the names of the source data files (e.g., `acoustic_example xxxx.asd`, where `xxxx` is the global time-step index of the transient solution) and an index file (e.g., `acoustic_example.index`) that will store the information associated with the source data. The **Write Frequency** allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict. In most situations, however, you will want to save the source data at every time step and use the default value of 1.

Since acoustics calculations usually generate thousands of time steps of source data, you may want to split the data into several files. Specifying the **Number of Time Steps per File** allows you to write the source data into separate files for different simulation intervals, the duration of which (in terms of the number of transient flow time steps) is specified by you. For example, if you specify 100 for this parameter, each file will contain source data for an interval length of 100 time steps regardless of the write frequency.

You will find this feature useful if you want to use a selected number of source data files to compute the sound pressure rather than using all the data. For example, you may want to exclude an initial portion of the source data from your acoustics calculation because you may realize later that the flow field has not fully attained a statistically steady state.

After you click **Apply**, ANSYS FLUENT will create the index file (e.g., `acoustics_example.index`), which contains information about the source data.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

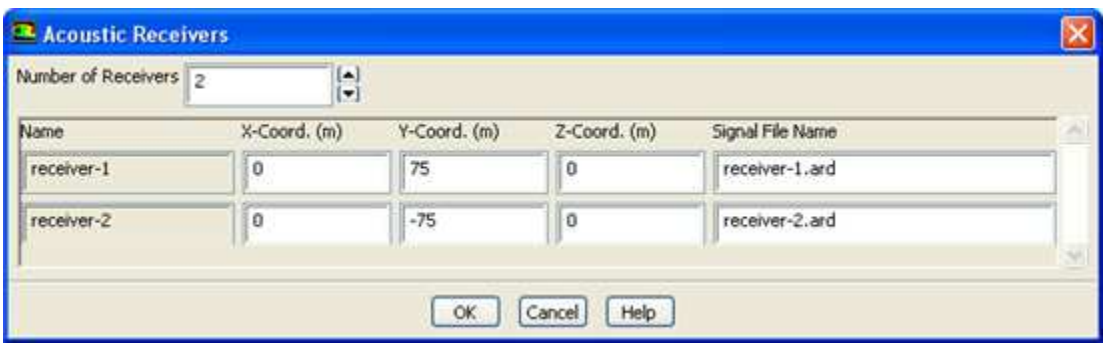
If you choose to save source data, keep in mind that the source data can use up a considerable amount of disk space, especially if the mesh being used has a large number of face elements on the source surfaces you selected. ANSYS FLUENT will print out the disk space requirement per time step at the time of source surface selection if the **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

At this point, if you have chosen to perform your acoustics calculation in two steps, (i.e., saving the source data first, and computing the sound at a later time), you can go ahead and instruct ANSYS FLUENT to perform a suitable number of time steps, and the source data will be saved to the disk. If you have chosen to perform an "on-the-fly" acoustic calculation, then you will need to specify receiver locations (see *Specifying Acoustic Receivers* (p. 998)) before you run the unsteady ANSYS FLUENT solution any further.

24.2.3. Specifying Acoustic Receivers

In the **Acoustics Model** dialog box, click the **Define Receivers...** button to open the **Acoustic Receivers** dialog box (*Figure 24.7* (p. 998)).

Figure 24.7 The Acoustic Receivers Dialog Box



Important

Note that you can also open the **Acoustic Receivers** dialog box by clicking the **Receivers...** button in the **Acoustic Sources** or the **Acoustic Signals** dialog box.

Increase the **Number of Receivers** to the total number of receivers for which you want to compute sound, and enter the coordinates for each receiver in the **X-Coord.**, **Y-Coord.**, and **Z-Coord.** fields. Note that because ANSYS FLUENT's acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (i.e., the selected source surfaces). The receiver locations can also fall outside of the computational domain.

For each receiver, you can specify a file name in the **Signal File Name** field. These files will be used to store the sound pressure signals at the corresponding receivers. By default, the files will be named `receiver-1.ard`, `receiver-2.ard`, etc.

Once the receiver locations have been defined, the setup for your acoustic calculation is complete.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.4. Specifying the Time Step

When using an implicit-in-time solution algorithm (dual-time stepping), and depending on the physical time step size and the most important time scales in the flow, you can write the acoustic source data at every time step. You can also coarsen it in time by a given frequency factor. The highest possible frequency the acoustic analysis can generate is based on the time step size of the collected acoustic source data.

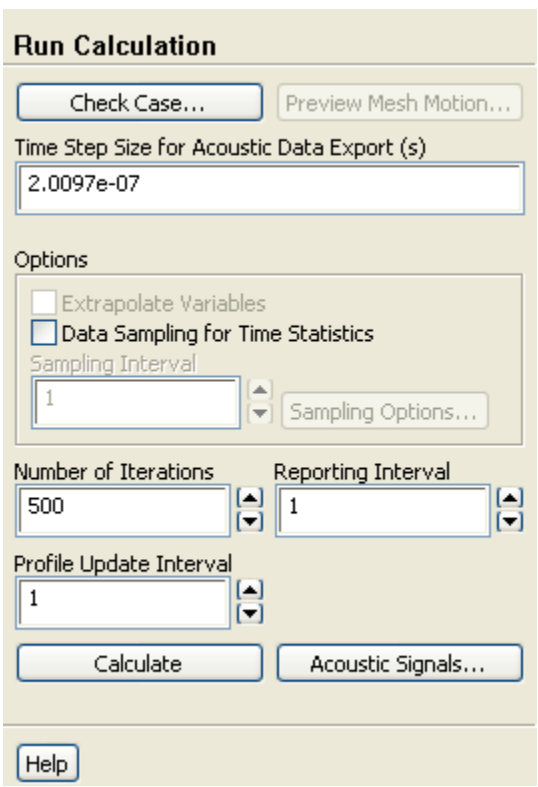
When using the density-based explicit solver, the physical time step must be computed by the solver, based on the CFL condition (Courant number). Due to the possibly large fluctuations of the physical time step, an adapting time-stepping procedure can be used when the FW-H acoustics model is enabled. This allows you to use a user-specified time interval for sampling the acoustic data. In turn, the solver adapts its time step, when necessary, without violating the CFL conditions to make sure that data is available at the desired time interval (hence, avoiding data interpolations).

In the **Run Calculation** task page (*Figure 24.8* (p. 999)), enter the **Time Step Size for Acoustic Data Export** to specify the time interval for acoustic data sampling. The value of this constant time step size determines the highest frequency that the acoustic analysis reproduces.

You can refer to *Performing Time-Dependent Calculations* (p. 1255) for more information about the **Run Calculation** task page.

Run Calculation

Figure 24.8 The Run Calculation Task Page



You can now proceed to instruct ANSYS FLUENT to perform a transient calculation for a suitable number of time steps. When the calculation is finished, you will have either the source data saved on files (if you chose to save it to a file or files), or the sound pressure signals (if you chose to perform an acoustic calculation "on

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the fly"), or both (if you chose to save the source data to files *and* if you chose to perform the acoustic calculation "on the fly").

If you chose to save the source data to files, the ANSYS FLUENT console window will print a message at the end of each time step indicating that source data have been written (or appended to) a file (e.g., `acoustic_example240.asd`).

24.2.5. Postprocessing the FW-H Acoustics Model Data

At this point, you will have either the source data saved to files or the sound pressure signals computed, or both. You can process these data to compute and plot various acoustic quantities using ANSYS FLUENT's FFT capabilities. See *Fast Fourier Transform (FFT) Postprocessing* (p. 1500) for more information.

24.2.5.1. Writing Acoustic Signals

If you chose to perform the acoustic calculation "on the fly", you will need to write the sound pressure data to files. To do so, select **Write Acoustic Signals** under **Options** in the **Acoustic Signals** dialog box (*Figure 24.9* (p. 1001)) and then click **Write**. The computed acoustic pressure will be saved from internal buffer memory into a separate file for each receiver you defined in the **Acoustic Receivers** dialog box (e.g., `receiver-1.ard`).

Run Calculation → **Acoustic Signals...**

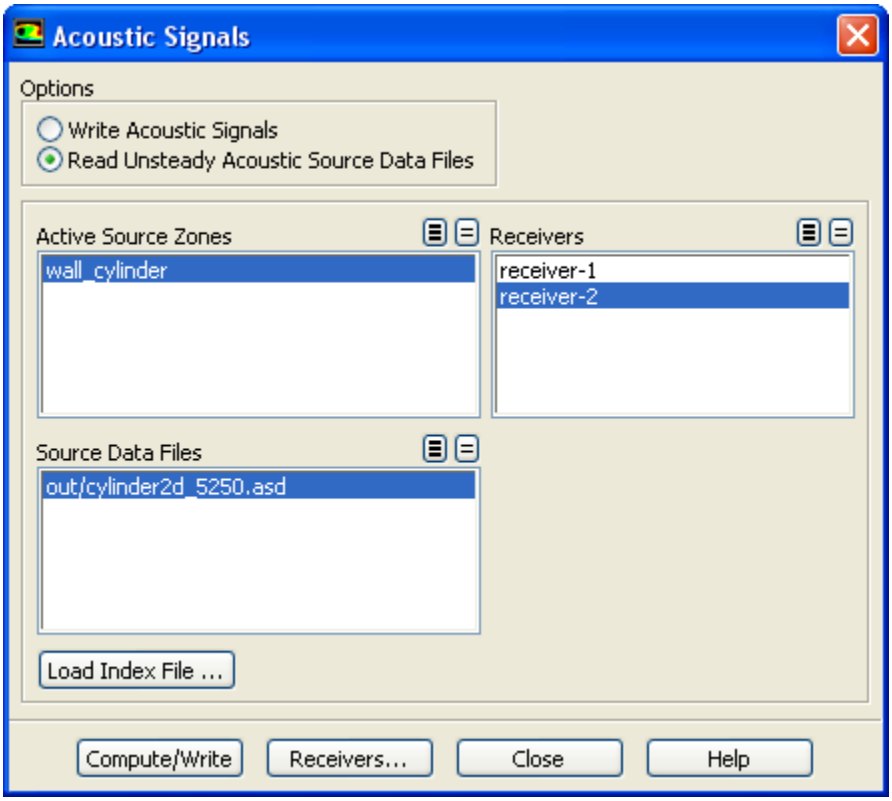
24.2.5.2. Reading Unsteady Acoustic Source Data

Computing the sound pressure signals using the source data saved to files is done in the **Acoustic Signals** dialog box (*Figure 24.9* (p. 1001))

Run Calculation → **Acoustic Signals...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.9 The Acoustic Signals Dialog Box



To compute the sound data, use the following procedure:

1. In the **Acoustic Signals** dialog box, select **Read Unsteady Acoustic Source Data Files** under **Options**.
2. Click **Load Index File...** and select the index file for your computation in the **Select File** dialog box. The file will have the name you entered in the **Source Data Root File Name** field in the **Acoustic Sources** dialog box, followed by the `.index` suffix (e.g., `acoustic_example.index`).
3. In the **Source Data Files** list, select the source data files that you want to use to compute sound. Source data files will all contain the specified root file name followed by the suffix `.asd`.

   Important

   You can use any number of source data files. However, note that you should select only consecutive files.

4. In the **Active Source Zones** list, select the source zones you want to include to compute sound. See *Specifying Source Surfaces* (p. 995) for details about proper source surface selection.
5. In the **Receivers** list, select the receivers for which you want to compute and save sound.

   Optionally, you can click the **Receivers...** button to open the **Acoustic Receivers** dialog box and define additional receivers.
6. Click the **Compute/Write** button to compute and save the sound pressure data. One file will be saved for each receiver you previously specified in the **Acoustic Receivers** dialog box (e.g., `receiver-1.ard`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you enabled both the **Export Acoustic Source Data in ASD Format** and **Compute Acoustic Signals Simultaneously** options in the **Acoustics Model** dialog box, you will need to first select the **Write Acoustic Signals** option in the **Acoustic Signals** dialog box after the flow simulation has been completed. If you select the **Read Unsteady Acoustic Source Data Files** before writing out the "on-the-fly" data in such a case, the data will be flushed out of the internal buffer memory. To avoid such a loss of data, you should save the ANSYS FLUENT case and data files whenever you begin to do an acoustic computation in the **Acoustic Signals** dialog box. The sound pressure data calculated "on the fly" will then be saved into the `.dat` file. Finally, after the "on-the-fly" data is saved, make sure to change the file names of the receivers before doing a sound pressure calculation with the **Read Unsteady Acoustic Source Data Files** option enabled, to avoid over-writing the "on-the-fly" signal files.

Important

Note that you can compute and write sound pressure signals only when the FW-H acoustics model has been enabled. See *Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS FLUENT ASD Format* (p. 993) for details about exporting source data (e.g., for SYSNOISE) without enabling the FW-H model.

24.2.5.2.1. Pruning the Signal Data Automatically

Before the computed sound pressure data at each receiver is saved, it is by default automatically pruned. Pruning of the receiver data means clipping the tails of the signal where incomplete source information is available.

The acoustic source data is tabulated from time τ_0 to τ_n. Without auto-pruning, the receiver register begins receiving the earliest sound pressure signal at

$$t_0 = \tau_0 + \frac{r_{min}}{a_0} \tag{24–1}$$

where r_{min} is the shortest distance between the source surfaces and the receiver. However, the receiver will not receive the sound pressure signal from the farthest point on the source surfaces (r_{max}) until the receiver time becomes

$$t_1 = \tau_0 + \frac{r_{max}}{a_0} \tag{24–2}$$

From time t_0 to t_1, the sound accumulated on the receiver register does not include the contribution from the entire source surface area, and thus the sound pressure data received during that time is not complete. The same thing occurs during the period from

$$t_m = \tau_m + \frac{r_{min}}{a_0} \tag{24–3}$$

to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$t_n = \tau_n + \frac{r_{max}}{a_0} \tag{24–4}$$

Thus, pruning means clipping the signal on the incomplete ends, from t_0 to t_1 and t_m to t_n. Auto-pruning can be disabled using the `define` → `models` → `acoustics` → `auto-prune` text command. Although auto-pruning can be disabled, it is expected that you will use only the complete sound pressure data.

24.2.5.3. Reporting the Static Pressure Time Derivative

The RMS value of the static pressure time derivative ($\partial p / \partial t$) is available for postprocessing only on wall surfaces, which are at the same time sources of sound, when the FW-H acoustics model is used.

You can select **Surface dpdt RMS** in the **Acoustics...** category only when you specify at least one wall surface, which is also marked as an acoustic source, in the relevant postprocessing dialog boxes.

24.2.5.4. Using the FFT Capabilities

Once the sound pressure signals are computed and saved in files, the sound data is ready to be analyzed using ANSYS FLUENT's FFT tools. In the **Fourier Transform** dialog box (*Figure 31.95* (p. 1503)), click on **Load Input File...** and select the appropriate `.ard` file. If the receiver data is still in ANSYS FLUENT's memory, then it can directly be processed using the **Process Receiver** option. See *Fast Fourier Transform (FFT) Post-processing* (p. 1500) for more information on ANSYS FLUENT's FFT capabilities.

Plots → **FFT** → **Set Up...**

24.3. Using the Broadband Noise Source Models

In this section, the procedure for setting up and using the broadband noise source models is outlined first, followed by descriptions of each of the steps involved.

The general procedure for carrying out a broadband noise source calculation in ANSYS FLUENT is as follows:

1. Calculate a steady or unsteady RANS solution.
2. Enable the broadband noise model and set the associated model parameters.

   Models → **Acoustics** → **Edit...**
3. Postprocess the noise sources.

   Graphics and Animations → **Contours** → **Set Up...**

For additional information, please see the following sections:

24.3.1. Enabling the Broadband Noise Source Models
24.3.2. Postprocessing the Broadband Noise Source Model Data

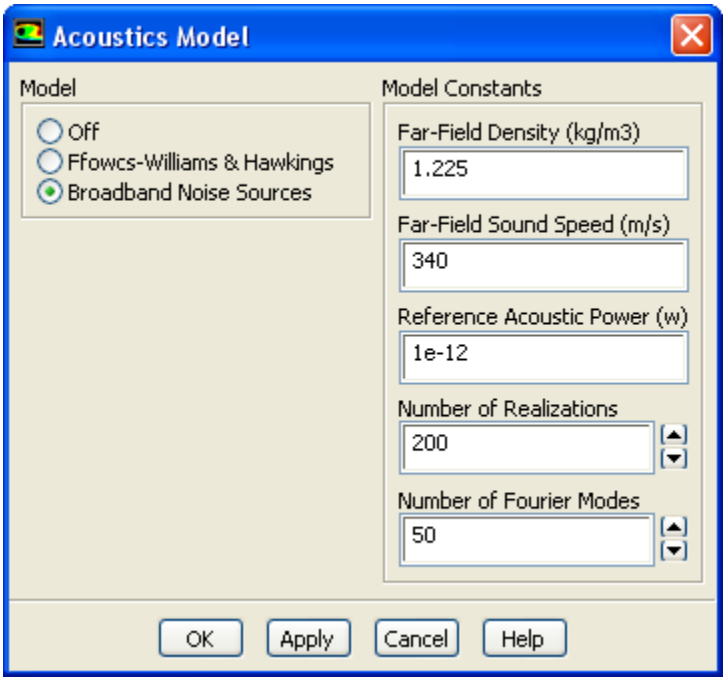
24.3.1. Enabling the Broadband Noise Source Models

To enable the broadband noise sources models, select **Broadband Noise Sources** in the **Acoustics Model** dialog box (*Figure 24.10* (p. 1004)).

Models → **Acoustics** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.10 The Acoustics Model Dialog Box for Broadband Noise



24.3.1.1. Setting Model Constants

Under **Model Constants** in the **Acoustics Model** dialog box, specify the relevant acoustic parameters and constants used by the model. See *Enabling the FW-H Acoustics Model* (p. 990) for the definitions of **Far-Field Density** and **Far-Field Sound Speed**.

Reference Acoustic Power

(for example, P_{ref} in Equation 15–13 in the Theory Guide) is used to compute the acoustic power outputs in decibels (dB). The default value is 10^{-12}. Note that the units for the reference acoustic power will be different in 2D (W/m^2) and 3D (W/m^3) cases.

Number of Realizations

is the number of samples used in the SNGR to compute the averaged source terms of LEE and Lilley's equations. The default value is 200.

Number of Fourier Modes

(N in Equation 15–33 in the Theory Guide) is the number of the Fourier modes used to compute the turbulent velocity field and its derivatives. The turbulent velocity field is then used to compute the LEE and Lilley's source terms. The default value is 50.

24.3.2. Postprocessing the Broadband Noise Source Model Data

The final step in the broadband noise source modeling process is the postprocessing of acoustic power and noise source data. The following variables are available in the **Acoustics...** postprocessing category:

- **Acoustic Power Level (dB)**
- **Acoustic Power**
- **Jet Acoustic Power Level (dB)** (axisymmetric models only)
- **Jet Acoustic Power** (axisymmetric models only)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Surface Acoustic Power Level (dB)**
- **Surface Acoustic Power**
- **Lilley's Self-Noise Source**
- **Lilley's Shear-Noise Source**
- **Lilley's Total Noise Source**
- **LEE Self-Noise X-Source**
- **LEE Shear-Noise X-Source**
- **LEE Total Noise X-Source**
- **LEE Self-Noise Y-Source**
- **LEE Shear-Noise Y-Source**
- **LEE Total Noise Y-Source**
- **LEE Self-Noise Z-Source** (3D models only)
- **LEE Shear-Noise Z-Source** (3D models only)
- **LEE Total Noise Z-Source** (3D models only)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 25: Modeling Discrete Phase

This chapter describes how to use the Lagrangian discrete phase capabilities available in ANSYS FLUENT. For information about the theory behind the discrete phase models, see "Discrete Phase" in the Theory Guide. Information is organized into the following sections:

25.1. Introduction
25.2. Steps for Using the Discrete Phase Models
25.3. Setting Initial Conditions for the Discrete Phase
25.4. Setting Boundary Conditions for the Discrete Phase
25.5. Setting Material Properties for the Discrete Phase
25.6. Solution Strategies for the Discrete Phase
25.7. Postprocessing for the Discrete Phase
25.8. Parallel Processing for the Discrete Phase Model

25.1. Introduction

In addition to solving transport equations for the continuous phase, ANSYS FLUENT allows you to simulate a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets or bubbles) dispersed in the continuous phase. ANSYS FLUENT computes the trajectories of these discrete phase entities, as well as heat and mass transfer to/from them. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included.

ANSYS FLUENT provides the following discrete phase modeling options:

- calculation of the discrete phase trajectory using a Lagrangian formulation that includes the discrete phase inertia, hydrodynamic drag, and the force of gravity, for both steady and unsteady flows
- prediction of the effects of turbulence on the dispersion of particles due to turbulent eddies present in the continuous phase
- heating/cooling of the discrete phase
- vaporization and boiling of liquid droplets
- combusting particles, including volatile evolution and char combustion to simulate coal combustion
- optional coupling of the continuous phase flow field prediction to the discrete phase calculations
- droplet breakup and coalescence
- consideration of particle/particle collisions and voidage of discrete phase

These modeling capabilities allow ANSYS FLUENT to simulate a wide range of discrete phase problems including particle separation and classification, spray drying, aerosol dispersion, bubble stirring of liquids, liquid fuel combustion, and coal combustion. The physical equations used for these discrete phase calculations are described in Particle Motion Theory – Laws for Heat and Mass Exchange in the Theory Guide, and instructions for setup, solution, and postprocessing are provided in *Steps for Using the Discrete Phase Models* (p. 1009) – *Postprocessing for the Discrete Phase* (p. 1067).

For additional information, please see the following sections:

25.1.1. Overview

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.1.2. Limitations

25.1.1. Overview

Currently, there are three approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach (discussed in Introduction in the Theory Guide), the Euler-Euler approach (discussed in Approaches to Multiphase Modeling), and the Dense Discrete Phase Model which is a hybrid Euler-Euler and Euler-Lagrange approach (discussed in Dense Discrete Phase Model in the Theory Guide).

25.1.2. Limitations

25.1.2.1. Limitation on the Particle Volume Fraction

The discrete phase formulation used by ANSYS FLUENT contains the assumption that the second phase is sufficiently dilute that particle-particle interactions and the effects of the particle volume fraction on the gas phase are negligible. In practice, these issues imply that the discrete phase must be present at a fairly low volume fraction, usually less than 10–12%. Note that the mass loading of the discrete phase may greatly exceed 10–12%: you may solve problems in which the mass flow of the discrete phase equals or exceeds that of the continuous phase. See *Modeling Multiphase Flows* (p. 1091) for information about when you might want to use one of the general multiphase models instead of the discrete phase model.

25.1.2.2. Limitation on Modeling Continuous Suspensions of Particles

The steady-particle Lagrangian discrete phase model is suited for flows in which particle streams are injected into a continuous phase flow with a well-defined entrance and exit condition. The Lagrangian model does not effectively model flows in which particles are suspended indefinitely in the continuum, as occurs in solid suspensions within closed systems such as stirred tanks, mixing vessels, or fluidized beds. The unsteady-particle discrete phase model, however, is capable of modeling continuous suspensions of particles. See *Modeling Multiphase Flows* (p. 1091) for information about when you might want to use one of the general multiphase models instead of the discrete phase models.

25.1.2.3. Limitations on Using the Discrete Phase Model with Other ANSYS FLUENT Models

The following restrictions exist on the use of other models with the discrete phase model:

- When tracking particles in parallel, the DPM model cannot be used with any of the multiphase flow models (VOF, mixture, or Eulerian – see *Modeling Multiphase Flows* (p. 1091)) if the shared memory option is enabled (*Parallel Processing for the Discrete Phase Model* (p. 1087)). (Note that using the message passing or hybrid option, when running in parallel, enables the compatibility of all multiphase flow models with the DPM model.)
- When using the DPM model with the Eulerian multiphase model, the tracked particles rely only on the primary phase to compute drag, heat, and mass transfer. Also, any DPM related source terms are applied to the primary phase. Particle tracking relative to a secondary phase is not provided.
- Streamwise periodic flow (either specified mass flow rate or specified pressure drop) cannot be modeled when the discrete phase model is used.
- Only nonreacting particles can be included when the premixed combustion model is used.
- Surface injections will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used, however only those surfaces associated with a boundary will be recalculated. Injections from cut plane surfaces will not be moved with the mesh and will be deleted when remeshing occurs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The cloud model is not available for unsteady particle tracking, or in parallel, when using the message passing or hybrid option for the particles.
- The wall-film model is only valid for liquid materials. If a nonliquid particle interacts with a wall-film boundary, the boundary condition will default to the reflect boundary condition.
- When multiple reference frames are used in conjunction with the discrete phase model, the display of particle tracks will not, by default, be meaningful. Similarly, coupled discrete-phase calculations are not meaningful.

  An alternative approach for particle tracking and coupled discrete-phase calculations with multiple reference frames is to track particles based on absolute velocity instead of relative velocity. To make this change, use the `define/models/dpm/ options/track-in-absolute-frame` text command. Note that the results may strongly depend on the location of walls inside the multiple reference frame.

  The particle injection velocities (specified in the **Set Injection Properties** dialog box) are defined relative to the frame of reference in which the particles are tracked. By default, the injection velocities are specified relative to the local reference frame. If you enable the `track-in-absolute-frame` option, the injection velocities are specified relative to the absolute frame.
- Relative particle tracking cannot be used in combination with sliding and moving deforming meshes. If sliding and/or deforming meshes are used with the DPM model, the particles will always be tracked in the absolute frame. Switching to the relative frame is not permitted.
- Boundedness of planes is not considered during sampling of particle tracks, which means that all particle tracks crossing the unbounded plane are sampled.

25.2. Steps for Using the Discrete Phase Models

You can include a discrete phase in your ANSYS FLUENT model by defining the initial position, velocity, size, and temperature of individual particles. These initial conditions, along with your inputs defining the physical properties of the discrete phase, are used to initiate trajectory and heat/mass transfer calculations. The trajectory and heat/mass transfer calculations are based on the force balance on the particle and on the convective/radiative heat and mass transfer from the particle, using the local continuous phase conditions as the particle moves through the flow. The predicted trajectories and the associated heat and mass transfer can be viewed graphically and/or alphanumerically.

The procedure for setting up and solving a problem involving a discrete phase is outlined below, and described in detail in *Options for Interaction with the Continuous Phase* (p. 1010) – *Postprocessing for the Discrete Phase* (p. 1067). Only the steps related specifically to discrete phase modeling are shown here. For information about inputs related to other models that you are using in conjunction with the discrete phase models, see the appropriate sections for those models.

1. Enable any of the discrete phase modeling options, if relevant, as described in *Options for Interaction with the Continuous Phase* (p. 1010).
2. Choose a transient or steady treatment of particles as described in *Steady/Transient Treatment of Particles* (p. 1010).
3. Specify tracking parameters and select a drag law as described in *Tracking Parameters for the Discrete Phase Model* (p. 1012).
4. Enable the required physical submodels for the discrete phase model, as described in *Physical Models for the Discrete Phase Model* (p. 1015).
5. Set the numerics parameters and solve the problem, as described in *Numerics of the Discrete Phase Model* (p. 1019) *Solution Strategies for the Discrete Phase* (p. 1063).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Specify the initial conditions and particle size distributions in injections, as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022).
7. Define the boundary conditions, as described in *Setting Boundary Conditions for the Discrete Phase* (p. 1050).
8. Define the material properties, as described in *Setting Material Properties for the Discrete Phase* (p. 1054).
9. Initialize the flow field.
10. Solve the coupled or uncoupled flow (*Solution Strategies for the Discrete Phase* (p. 1063)).
11. For transient cases, advance the solution in time by taking the desired number of time steps. Particle positions will be updated as the solution advances in time. If you are solving an uncoupled flow, the particle position will be updated at the end of each time step. For a coupled calculation, the positions are iterated on or within each time step.
12. Examine the results, as described in *Postprocessing for the Discrete Phase* (p. 1067).

25.2.1. Options for Interaction with the Continuous Phase

If the discrete phase interacts (i.e., exchanges mass, momentum, and/or energy) with the continuous phase, you should enable the **Interaction with Continuous Phase** option in the **Discrete Phase Model** dialog box (*Figure 25.1* (p. 1013)).

Models → **Discrete Phase** → **Edit...**

An input for the **Number of Continuous Phase Iterations per DPM Iteration** will appear, which allows you to control the frequency at which the particles are tracked and the DPM sources are updated.

For steady-state simulations, increasing the **Number of Continuous Phase Iterations per DPM Iteration** will increase stability but require more iterations to converge.

In addition, another option exists which allows you to control the numerical treatment of the source terms and how they are applied to the continuous phase equations. **Update DPM Sources Every Flow Iteration** is recommended when doing unsteady simulations; at every DPM Iteration, the particle source terms are recalculated. The source terms applied to the continuous phase equations transition to the new values every flow iteration based on Equation 16–305 to Equation 16–307 in the Theory Guide. This process is controlled by the under-relaxation factor, specified in the **Solution Controls** task page, see *Under-Relaxation of the Interphase Exchange Terms* (p. 1066).

25.2.2. Steady/Transient Treatment of Particles

The Discrete Phase Model utilizes a Lagrangian approach to derive the equations for the underlying physics which are solved transiently. Transient numerical procedures in the Discrete Phase Model can be applied to resolve steady flow simulations as well as transient flows.

In the *Discrete Phase Model Dialog Box* (p. 1723) you have the option of choosing whether you want to treat the particles in an unsteady or a steady fashion. This option can be chosen independent of the settings for the solver. Thus, you can perform steady state trajectory simulations even when selecting a transient solver for numerical reasons. You can also specify unsteady particle tracking when solving the steady continuous phase equations. This can be used to improve numerical stability for very large particle source terms or simply for postprocessing purposes. Whenever you enable a breakup or collision model to simulate sprays, the **Unsteady Particle Tracking** will be switched on automatically.

When **Unsteady Particle Tracking** is enabled, several new options appear. If steady state equations are solved for the continuous phase, you simply enter the **Particle Time Step Size** and the **Number of Time**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps, thus tracking particles every time a DPM iteration is conducted. When you increase the **Number of Time Steps**, the droplets penetrate the domain faster.

When solving unsteady equations for the continuous phase, you must decide whether you want to use **Fluid Flow Time Step** to inject the particles, or whether you prefer a **Particle Time Step Size** independent of the **Fluid Flow Time Step**. With the latter option, you can use the Discrete Phase Model in combination with changes in the time step for the continuous equations, as it is done when using adaptive flow time stepping.

If you do not use **Fluid Flow Time Step**, you will need to decide when to inject the particles for a new time step. You can either **Inject Particles at Particle Time Step** or at the **Fluid Flow Time Step**. In any case, the particles will always be tracked in such a way that they coincide with the flow time of the continuous flow solver.

You can use a user-defined function (`DEFINE_DPM_TIMESTEP`) to change the time step for DPM particle tracking. The time step can be prescribed for special applications where a certain time step is needed. For more information about changing the time step size for DPM particle tracking, see DEFINE_DPM_TIMESTEP in the UDF Manual.

Important

When the density-based solver is used with the explicit unsteady formulation, the particles are advanced once per time step and are calculated at the start of the time step (before the flow is updated).

Additional inputs are required for each injection in the *Set Injection Properties Dialog Box* (p. 2097), as detailed in *Defining Injection Properties* (p. 1040). For **Unsteady Particle Tracking**, the injection **Start Time** and **Stop Time** must be specified under **Point Properties**. Injections with start and stop times set to zero will be injected only at the start of the calculation ($t=0$). If the In-Cylinder mesh motion is enabled, the start and stop times are replaced by **Start Crank Angle** and **Stop Crank Angle**, respectively. The injection specified in this way will be repeated at the starting and stopping crank angle if the simulation is run through more than one cycle. Changing injection settings during a transient simulation will not affect particles currently released in the domain. At any point during a simulation, you can clear particles that are currently in the domain by clicking the **Clear Particles** button in the *Discrete Phase Model Dialog Box* (p. 1723).

For transient simulations, several methods can be chosen to control when the particles are advanced.

- If the **Number of Continuous Phase Iterations per DPM Iteration** is less than the number of iterations required to converge the continuous phase between time steps, then sub-iterations are done. Here, particles are tracked to their new positions during a time step and DPM sources are updated; particles are then returned to their original state at the beginning of the time step. At the end of the time step, particles are advanced to their new positions based on the continuous-phase solution.
- If the **Number of Continuous Phase Iterations per DPM Iteration** is larger than the number of iterations specified to converge the continuous phase between time steps, the particles are advanced at the beginning of the time step to compute the particle source terms.
- When you specify a value of zero as the **Number of Continuous Phase Iterations per DPM Iteration**, the particles are advanced at the end of the time step. For this option, it may be better if the particle source terms are not reset at the beginning of the time step. This can be done with the TUI command `define/models/dpm/interaction/reset-sources-at-timestep?`.

In all the above cases, you must provide a sufficient number of particle source term updates to better control when the particles are advanced, see *Figure 25.26* (p. 1067).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

In steady-state discrete phase modeling, particles do not interact with each other and are tracked one at a time in the domain.

Important

If the collision model is used, you will not be able to set the **Number of Continuous Phase Iterations per DPM Iteration**. Refer to Droplet Collision and Coalescence Model Theory in the Theory Guide for details about this limitation.

25.2.3. Tracking Parameters for the Discrete Phase Model

You will use two parameters to control the time integration of the particle trajectory equations:

- the maximum number of time steps

  This factor is used to abort trajectory calculations when the particle never exits the flow domain.
- the length scale/step length factor

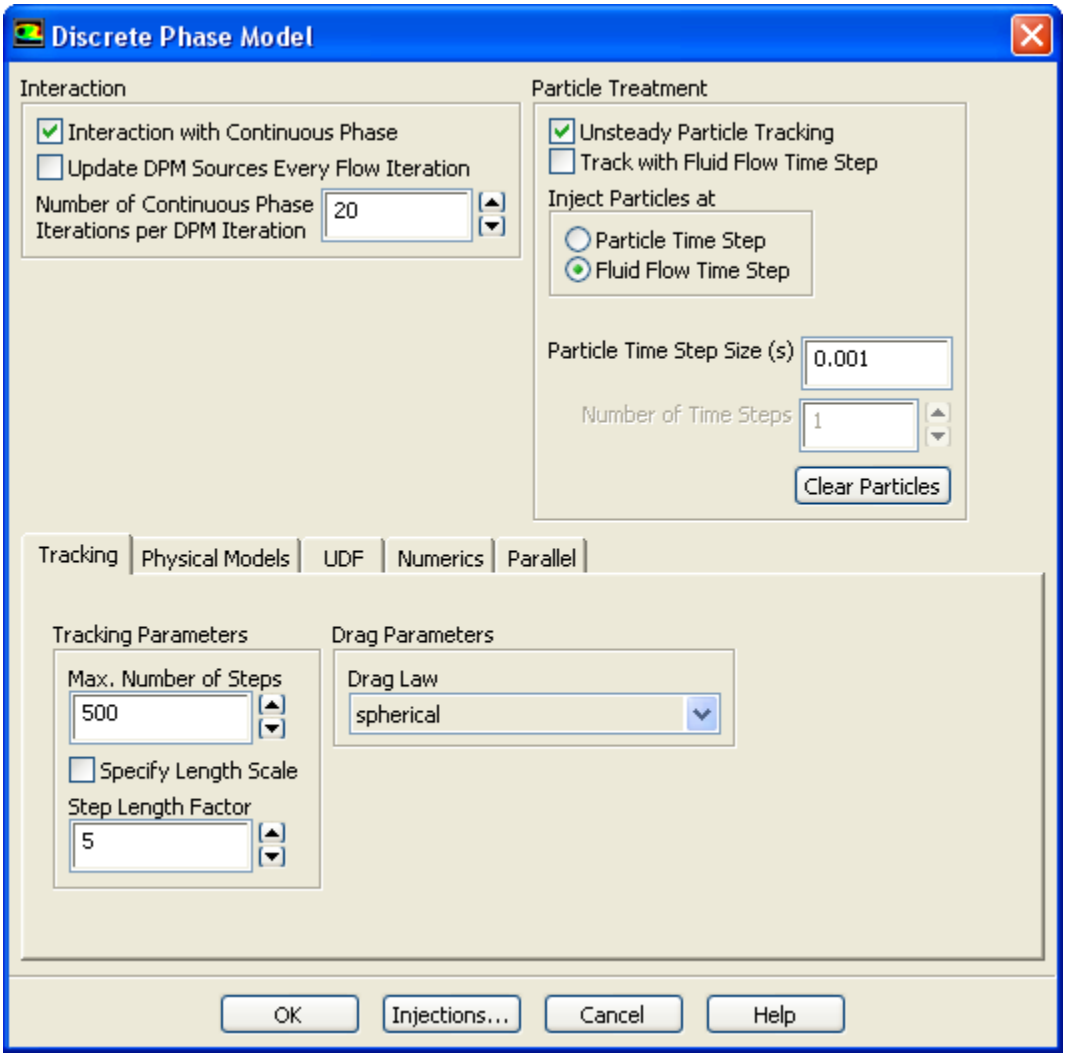
  This factor is used to set the time step for integration within each control volume.

Each of these parameters is set in the *Discrete Phase Model Dialog Box* (p. 1723) (*Figure 25.1* (p. 1013)) under **Tracking Parameters** in the **Tracking** tab.

Models → **Discrete Phase** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.1 The Discrete Phase Model Dialog Box and the Tracking Parameters



Max. Number of Steps

is the maximum number of time steps used to compute a single particle trajectory via integration of Equation 16–1 in the Theory Guide. When the maximum number of steps is exceeded, ANSYS FLUENT abandons the trajectory calculation for the current particle injection and reports the trajectory fate as "incomplete". The limit on the number of integration time steps eliminates the possibility of a particle being caught in a recirculating region of the continuous phase flow field and being tracked infinitely. Note that you may easily create problems in which the default value of 500 time steps is insufficient for completion of the trajectory calculation. In this case, when trajectories are reported as incomplete within the domain and the particles are not recirculating indefinitely, you can increase the maximum number of steps (up to a limit of 10^9).

Length Scale

controls the integration time step size used to integrate the equations of motion for the particle. The integration time step is computed by ANSYS FLUENT based on a specified length scale L, and the velocity of the particle (u_p) and of the continuous phase (u_c):

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta t = \frac{L}{u_p + u_c} \tag{25–1}$$

where L is the **Length Scale** that you define. As defined by *Equation 25–1* (p. 1014), L is proportional to the integration time step and is equivalent to the distance that the particle will travel before its motion equations are solved again and its trajectory is updated. A smaller value for the **Length Scale** increases the accuracy of the trajectory and heat/mass transfer calculations for the discrete phase.

(Note that particle positions are always computed when particles enter/leave a cell; even if you specify a very large length scale, the time step used for integration will be such that the cell is traversed in one step.)

Length Scale will appear in the **Discrete Phase Model** dialog box when the **Specify Length Scale** option is enabled.

Step Length Factor

also controls the time step size used to integrate the equations of motion for the particle. It differs from the **Length Scale** in that it allows ANSYS FLUENT to compute the time step in terms of the number of time steps required for a particle to traverse a computational cell. To set this parameter instead of the **Length Scale**, turn off the **Specify Length Scale** option.

The integration time step is computed by ANSYS FLUENT based on a characteristic time that is related to an estimate of the time required for the particle to traverse the current continuous phase control volume. If this estimated transit time is defined as Δt^*, ANSYS FLUENT chooses a time step Δt as

$$\Delta t = \frac{\Delta t^*}{\lambda} \tag{25–2}$$

where λ is the **Step Length Factor**. As defined by *Equation 25–2* (p. 1014), λ is inversely proportional to the integration time step and is roughly equivalent to the number of time steps required to traverse the current continuous phase control volume. A larger value for the **Step Length Factor** decreases the discrete phase integration time step. The default value for the **Step Length Factor** is 5. **Step Length Factor** will appear in the **Discrete Phase Model** dialog box when the **Specify Length Scale** option is off (the default setting).

One simple rule of thumb to follow when setting the parameters above is that if you want the particles to advance through a domain consisting of N mesh cells into the main flow direction, the **Step Length Factor** times N should be approximately equal to the **Max. Number of Steps**.

When **Accuracy Control** is activated in the **Numerics** tab, the settings for **Step Length Factor** and **Length Scale** will be used only to estimate the time step of the first integration step. In all subsequent integration steps, the particle integration time step is adapted to achieve the tolerance specified in *Numerics of the Discrete Phase Model* (p. 1019).

25.2.4. Drag Laws

There are eight drag laws for the particles that can be selected from the **Drag Law** drop-down list under **Drag Parameters**.

The **spherical**, **nonspherical**, **Stokes-Cunningham**, and **high-Mach-number** laws described in Particle Force Balance in the Theory Guide are always available, and the **dynamic-drag** law described in Dynamic Drag

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Theory in the Theory Guide is available only when one of the droplet breakup models is used in conjunction with unsteady tracking. See *Modeling Spray Breakup* (p. 1017) for information about enabling the droplet breakup models. The remaining three, **Wen-Yu**, **Gidaspow**, and **Syamlal-OBrien** are available only when the dense discrete phase model is enabled (*Including the Dense Discrete Phase Model* (p. 1167)) and the flow regime consists of a dense gas-solid. However, with these models, you cannot verify up front whether it really is a dense flow or a gas-solid flow indeed. It is up to you to decide. In any case, these drag formulations are suitable for dense gas-solid flows.

If the **spherical**, **high-Mach-number**, **dynamic-drag**, **Wen-Yu**, **Gidaspow**, or **Syamlal-OBrien** drag law is selected, no further inputs are required. If the **nonspherical** law is selected, the particle **Shape Factor** (ϕ in Equation 16–66 in the Theory Guide) must be specified. The shape factor value cannot exceed 1. For the **Stokes-Cunningham** law, the **Cunningham Correction** factor (C_c in Equation 16–68 in the Theory Guide) must be specified.

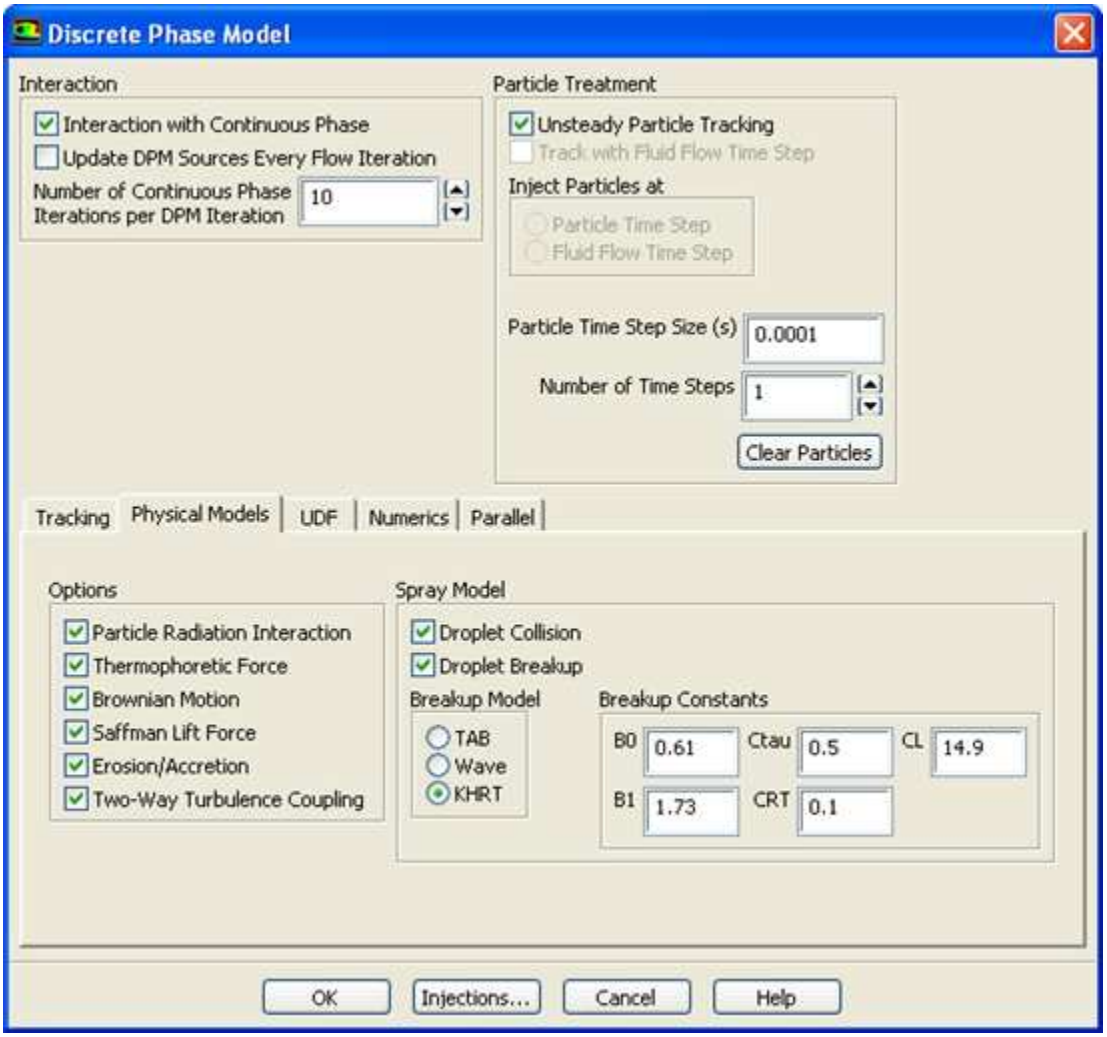
25.2.5. Physical Models for the Discrete Phase Model

This section provides instructions for using the optional discrete phase models available in ANSYS FLUENT. All of them can be enabled in the **Physical Models** tab of the **Discrete Phase Model** dialog box (*Figure 25.2* (p. 1016)).

Models → **Discrete Phase** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.2 The Discrete Phase Model Dialog Box and the Physical Models



25.2.5.1. Including Radiation Heat Transfer Effects on the Particles

If you want to include the effect of radiation heat transfer to the particles (Equation 5–34 in the Theory Guide), you must enable the **Particle Radiation Interaction** option under the **Physical Models** tab, in the *Discrete Phase Model Dialog Box* (p. 1723) (*Figure 25.2* (p. 1016)). You will also need to define additional properties for the particle materials (emissivity and scattering factor), as described in *Description of the Properties* (p. 1059). This option is available only when the P-1 or discrete ordinates radiation model is used.

25.2.5.2. Including Thermophoretic Force Effects on the Particles

If you want to include the effect of the thermophoretic force on the particle trajectories (Equation 16–8 in the Theory Guide), enable the **Thermophoretic Force** option under the **Physical Models** tab, in the *Discrete Phase Model Dialog Box* (p. 1723). You will also need to define the thermophoretic coefficient for the particle material, as described in *Description of the Properties* (p. 1059).

25.2.5.3. Including Brownian Motion Effects on the Particles

For sub-micron particles in laminar flow, you may want to include the effects of Brownian motion (described in Brownian Force in the Theory Guide) on the particle trajectories. To do so, enable the **Brownian Motion** option under the **Physical Models** tab. When Brownian motion effects are included, it is recommended that

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you also select the **Stokes-Cunningham** drag law in the **Drag Law** drop-down list under **Drag Parameters**, and specify the **Cunningham Correction** (C_c in Equation 16–68 in the Theory Guide).

25.2.5.4. Including Saffman Lift Force Effects on the Particles

For sub-micron particles, you can also model the lift due to shear (the Saffman lift force, described in Saffman's Lift Force in the Theory Guide) in the particle trajectory. To do this, enable the **Saffman Lift Force** option under the **Physical Models** tab, in the *Discrete Phase Model Dialog Box* (p. 1723).

25.2.5.5. Monitoring Erosion/Accretion of Particles at Walls

Particle erosion and accretion rates can be monitored at wall boundaries. These rate calculations can be enabled in the *Discrete Phase Model Dialog Box* (p. 1723) when the discrete phase is coupled with the continuous phase (i.e., when **Interaction with Continuous Phase** is selected). Enabling the **Erosion/Accretion** option will cause the erosion and accretion rates to be calculated at wall boundary faces when particle tracks are updated. You will also need to set the **Impact Angle Function** ($f(\alpha)$ in Equation 16–205 in the Theory Guide), **Diameter Function** ($C\left(d_p\right)$ in Equation 16–205), and **Velocity Exponent Function** ($b(v)$ in Equation 16–205) in the **Wall** boundary conditions dialog box for each wall zone (as described in *Discrete Phase Boundary Condition Types* (p. 1052)).

25.2.5.6. Including the Effect of Particles on Turbulent Quantities

Particles can damp or produce turbulent eddies [*57*] (p. 2212). In ANSYS FLUENT, the work done by the turbulent eddies on the particles is subtracted from the turbulent kinetic energy using the formulation described in [*22*] (p. 2210) and [*6*] (p. 2209).

If you want to consider these effects in the chosen turbulence model, you can enable this using **Two-Way Turbulence Coupling**, under the **Physical Models** tab.

25.2.6. Options for Spray Modeling

Select the spray model by going to the **Physical Models** tab in **Discrete Phase Model** dialog box. Two spray models are available: **Droplet Collision** and **Droplet Breakup**.

25.2.6.1. Modeling Droplet Collisions

To include the effect of droplet collisions, as described in Droplet Collision and Coalescence Model Theory in the Theory Guide, select the **Droplet Collision** option under **Spray Models**. There are no further inputs for this model.

25.2.6.2. Modeling Spray Breakup

To enable the modeling of spray breakup, select the **Droplet Breakup** option under **Spray Model** and then select the desired model (**TAB**, **Wave**, or **KHRT**). A detailed description of these models can be found in Secondary Breakup Model Theory in the Theory Guide.

For the TAB model, you will need to specify a value for y_0 (the initial distortion at time equal to zero in Equation 16–255 in the Theory Guide) in the **y0** field. The default value ($y_0 = 0$) is recommended. You will also set the number of **Breakup Parcels** (under **Breakup Constants**), to split the droplet into several child parcels, as described in Velocity of Child Droplets in the Theory Guide. The diameter of the child parcels is sampled from a Rosin-Rammler distribution. This can be switched off in the TUI with the command

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/define/models/dpm/spray-modeling/randomize-tab-diameters?
```

For the Wave model, you will need to specify values for **B0** and **B1**, where **B0** is the constant B_0 in Equation 16–284 in the Theory Guide and **B1** is the constant B_1 in Equation 16–286. You will generally not need to modify the value of **B0**, as the default value 0.61 is acceptable for nearly all cases. A value of 1.73 is recommended for **B1**.

For the KHRT model, you will need to specify values for **B0**, **B1**, **Ctau**, **CRT**, and **CL**. The constants **B0** and **B1** are the same as for the Wave model. The other constants are explained in detail in KHRT Breakup Model in the Theory Guide.

For steady-state simulations, you will also need to specify an appropriate **Particle Time Step Size** and the **Number of Time Steps** which will control the spray density. See *Options for Interaction with the Continuous Phase* (p. 1010) for more information.

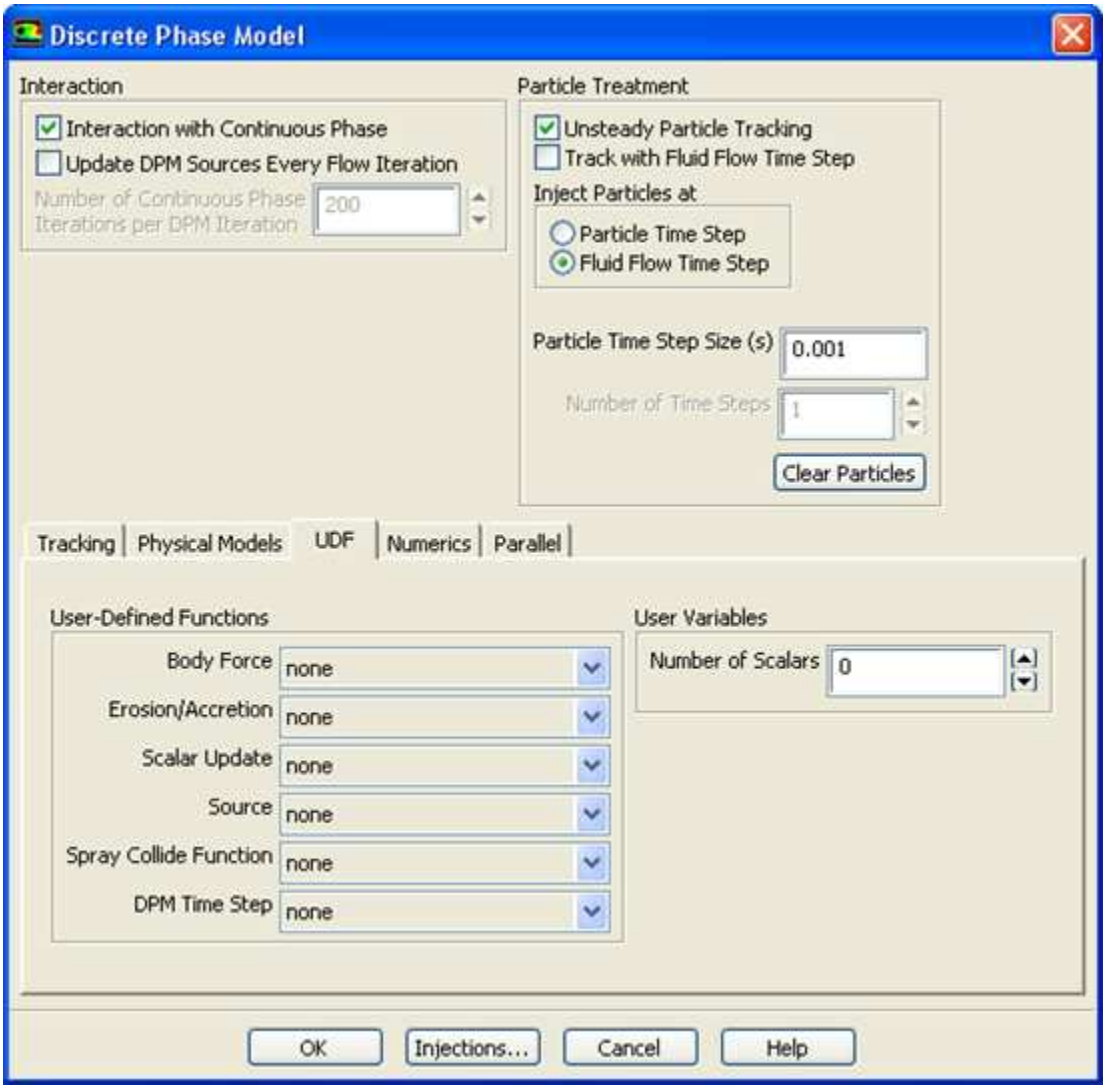
Note that you may want to use the dynamic drag law when you use one of the spray breakup models. See *Drag Laws* (p. 1014) for information about choosing the drag law.

25.2.7. User-Defined Functions

User-defined functions can be used to customize the discrete phase model to include additional body forces, modify interphase exchange terms (sources), calculate or integrate scalar values along the particle trajectory, and incorporate nonstandard erosion rate definitions. More information about user-defined functions can be found in the UDF Manual.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.3 The Discrete Phase Model Dialog Box and the UDFs



In the *Discrete Phase Model Dialog Box* (p. 1723), under **User-Defined Functions** in the **UDF** tab, there are drop-down lists labeled **Body Force**, **Scalar Update**, **Source**, **Spray Collide Function**, and **DPM Time Step** (*Figure 25.3* (p. 1019)). If **Erosion/Accretion** is enabled under the **Physical Models** tab, there will be an additional drop-down list labeled **Erosion/Accretion**. These lists will show available user-defined functions that can be selected to customize the discrete phase model.

In addition, you can specify a **Number of Scalars** which are allocated to each particle and can be used to store information when implementing your own particle models.

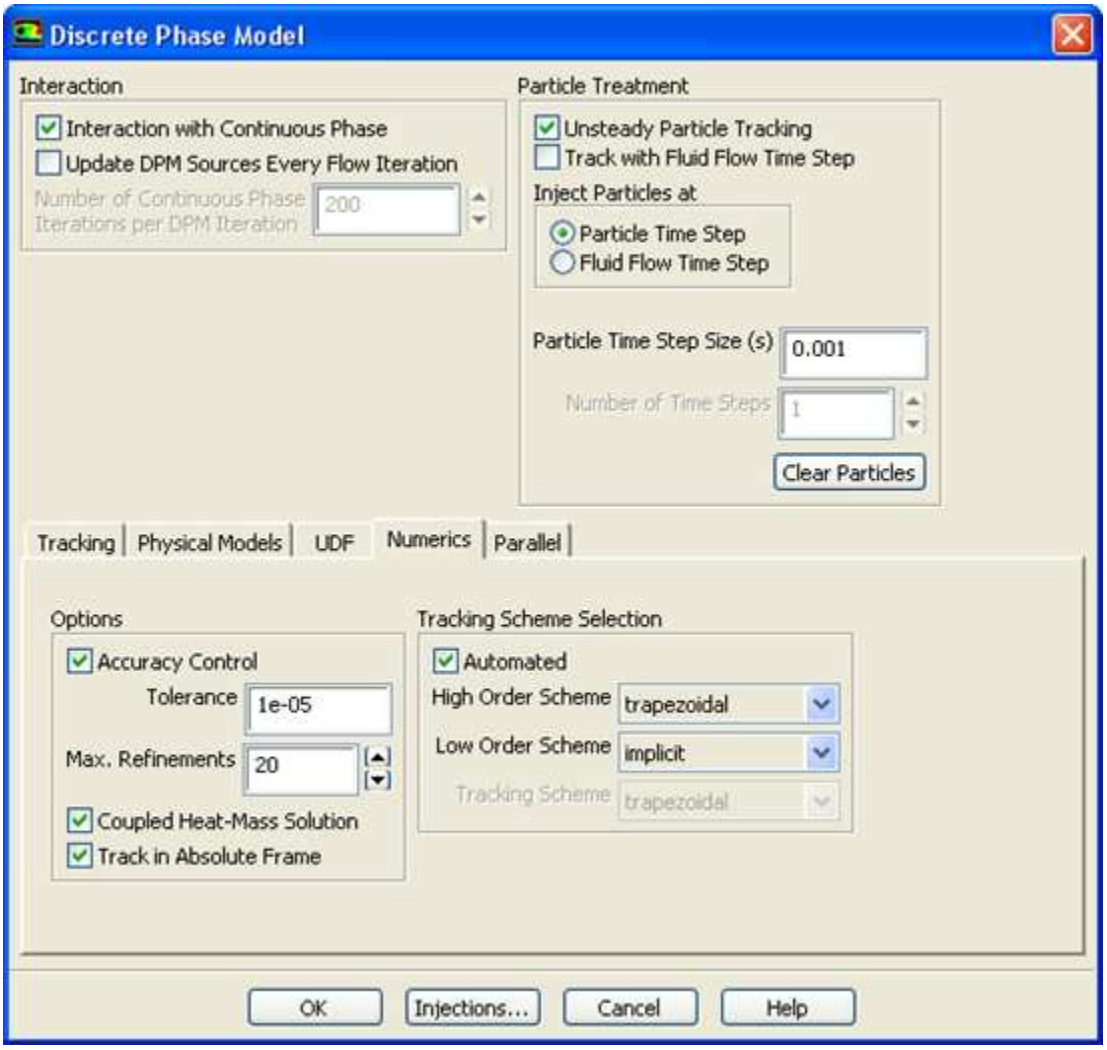
A user defined drag law needs to be selected in the **Tracking** tab.

25.2.8. Numerics of the Discrete Phase Model

The underlying physics of the Discrete Phase Model is described by ordinary differential equations (ODE) as opposed to the continuous flow which is expressed in the form of partial differential equations (PDE). Therefore, the Discrete Phase Model uses its own numerical mechanisms and discretization schemes, which are completely different from other numerics used in ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.4 The Discrete Phase Model Dialog Box and the Numerics



The **Numerics** tab gives you control over the numerical schemes for particle tracking as well as solutions of heat and mass equations (*Figure 25.4* (p. 1020)).

25.2.8.1. Numerics for Tracking of the Particles

To solve equations of motion for the particles, the following numerical schemes are available:

implicit
uses an implicit Euler integration of Equation 16–1 in the Theory Guide which is unconditionally stable for all particle relaxation times.

trapezoidal
uses a semi-implicit trapezoidal integration.

analytic
uses an analytical integration of Equation 16–1 of the Theory Guide where the forces are held constant during the integration.

runge-kutta
facilitates a 5th order Runge Kutta scheme derived by Cash and Karp [*13*] (p. 2209).

For additional details, see *Solution Strategies for the Discrete Phase* (p. 1063).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can either choose a single tracking scheme, or switch between higher order and lower order tracking schemes using an automated selection based on the accuracy to be achieved and the stability range of each scheme. In addition, you can control how accurately the equations need to be solved.

Accuracy Control
enables the solution of equations of motion within a specified tolerance. This is done by computing the error of the integration step and reducing the integration step if the error is too large. If the error is within the given tolerance, the integration step will also be increased in the next steps.

Tolerance
is the maximum relative error which has to be achieved by the tracking procedure. Based on the numerical scheme, different methods are used to estimate the relative error. The implemented Runge-Kutta scheme uses an embedded error control mechanism. The error of the other schemes is computed by comparing the result of the integration step with the outcome of a two step procedure with half the step size.

Max. Refinements
is the maximum number of step size refinements in one single integration step. If this number is exceeded the integration will be conducted with the last refined integration step size.

Automated Tracking Scheme Selection
provides a mechanism to switch in an automated fashion between numerically stable lower order schemes and higher order schemes, which are stable only in a limited range. In situations where the particle is far from hydrodynamic equilibrium, an accurate solution can be achieved very quickly with a higher order scheme, since these schemes need less step refinements for a certain tolerance. When the particle reaches hydrodynamic equilibrium, the higher order schemes become inefficient since their step length is limited to a stable range. In this case, the mechanism switches to a stable lower order scheme and facilitates larger integration steps.

Important

This mechanism is only available when **Accuracy Control** is enabled.

Higher Order Scheme
can be chosen from the group consisting of **trapezoidal** and **runge-kutta** scheme.

Lower Order Scheme
consists of **implicit**and the exponential **analytic** integration scheme.

Tracking Scheme
is selectable only if **Automated** is switched off. You can choose any of the tracking schemes. You also can combine each of the tracking schemes with **Accuracy Control**.

25.2.8.2. Including Coupled Heat-Mass Solution Effects on the Particles

By default, the solution of the particle heat and mass equations are solved in a segregated manner. If you enable the **Coupled Heat-Mass Solution** option, ANSYS FLUENT will solve this pair of equations using a stiff, coupled ODE solver with error tolerance control. The increased accuracy, however, comes at the expense of increased computational time.

25.2.8.3. Tracking in a Reference Frame

Particle tracking is related to a coordinate system. With **Track in Absolute Frame** enabled, you can choose to track the particles in the absolute reference frame. All particle coordinates and velocities are then computed in this frame. The forces due to friction with the continuous phase are transformed to this frame automatically.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In rotating flows it might be appropriate for numerical reasons to track the particles in the relative reference frame. If several reference frames exist in one simulation, then the particle velocities are transformed to each reference frame when they enter the fluid zone associated with this reference frame.

25.2.8.4. Staggering of Particles in Space and Time

In order to obtain a better representation of an injector, the particles can be *staggered* either spatially or temporally. When particles are staggered spatially, ANSYS FLUENT randomly samples from the region in which the spray is specified (e.g., the sheet thickness in the pressure-swirl atomizer) so that as the calculation progresses, trajectories will originate from the entire region. This allows the entire geometry specified in the atomizer to be sampled while specifying fewer streams in the input dialog box, thus decreasing computational expense.

When injecting particles in a transient calculation using relatively large time steps in relation to the spray event, the particles can clump together in discrete bunches. The clumps do not look physically realistic, though ANSYS FLUENT calculates the trajectory for each particle as it passes through a cell and the coupling to the gas phase is properly accounted for. To obtain a statistically smoother representation of the spray, the particles can be staggered in time. During the first time step, the particle is tracked for a random percentage of its initial step. This results in a sample of the initial volume swept out by the particle during the first time step and a smoother, more uniform spatial distribution at longer time intervals.

The menu for staggering is available in the text user interface (TUI), under

`define/models/dpm/options/particle-staggering`.

The "staggering factor" in the TUI is a constant which multiplies the random sample. The staggering factor controls the percentage of the initial time step that will be sampled. For example, if the staggering factor is 0.5, then the parcels in the injection will be tracked between half and all of their full initial time step. If the staggering factor is 0.1, then the parcels will be tracked between ninety percent and all of their initial time step. If the staggering factor is set to 0.9, the parcels will be tracked between ten percent and all of their initial time step. This allows you to control the amount of smoothing between injections.

The default values for the options in the TUI are no temporal staggering and a temporal staggering factor of 1.0. The temporal staggering factor is inactive until the flag for temporal staggering is enabled.

25.3. Setting Initial Conditions for the Discrete Phase

The primary inputs that you must provide for the discrete phase calculations in ANSYS FLUENT are the initial conditions that define the starting positions, velocities, and other parameters for each particle stream and the physical effects acting on the particle streams, requiring additional particle properties. You will define the initial conditions for a particle/droplet stream by creating an "injection" and assigning properties to it.

The required initial conditions depend on the injection type, while the physical effects are selected by choosing an appropriate particle type. For some injection types you can provide a particle size distribution, like the Rosin-Rammler distribution, see *Using the Rosin-Rammler Diameter Distribution Method* (p. 1035).

The initial conditions provide the starting values for all of the dependent discrete phase variables that describe the instantaneous conditions of an individual particle, and include the following:

- position (x, y, z coordinates) of the particle
- velocities (u, v, w) of the particle

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocity magnitudes and spray cone angle can also be used (in 3D) to define the initial velocities (see *Point Properties for Cone Injections* (p. 1027)). For moving reference frames, relative velocities should be specified.

- diameter of the particle, d_p
- temperature of the particle, T_p
- mass flow rate of the particle stream that will follow the trajectory of the individual particle/droplet, $\dot{m}_p$ (required only for coupled calculations)
- additional parameters if one of the atomizer models described in Atomizer Model Theory in the Theory Guide is used for the injection

Important

When an atomizer model is selected, you will not input initial diameter, velocity, and position quantities for the particles due to the complexities of sheet and ligament breakup. Instead of initial conditions, the quantities you will input for the atomizer models are global parameters.

These dependent variables (temperature, diameter, etc.) are updated according to the equations of motion (Particle Motion Theory in the Theory Guide) and according to the heat/mass transfer relations applied (Laws for Heat and Mass Exchange in the Theory Guide) as the particle/droplet moves along its trajectory. You can define any number of different sets of initial conditions for discrete phase particles/droplets provided that your computer has sufficient memory.

For the setup of transient particle cases, the point properties for mass flow and velocity can be specified using transient profiles (see *Point Properties for Transient Injections* (p. 1049)).

For additional information, please see the following sections:

25.3.1. Injection Types
25.3.2. Particle Types
25.3.3. Point Properties for Single Injections
25.3.4. Point Properties for Group Injections
25.3.5. Point Properties for Cone Injections
25.3.6. Point Properties for Surface Injections
25.3.7. Point Properties for Plain-Orifice Atomizer Injections
25.3.8. Point Properties for Pressure-Swirl Atomizer Injections
25.3.9. Point Properties for Air-Blast/Air-Assist Atomizer Injections
25.3.10. Point Properties for Flat-Fan Atomizer Injections
25.3.11. Point Properties for Effervescent Atomizer Injections
25.3.12. Point Properties for File Injections
25.3.13. Using the Rosin-Rammler Diameter Distribution Method
25.3.14. Creating and Modifying Injections
25.3.15. Defining Injection Properties
25.3.16. Specifying Turbulent Dispersion of Particles
25.3.17. Custom Particle Laws
25.3.18. Defining Properties Common to More than One Injection
25.3.19. Point Properties for Transient Injections

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.1. Injection Types

ANSYS FLUENT provides 11 types of injections:

- single
- group
- cone (only in 3D)
- solid-cone (only in 3D)
- surface
- plain-orifice atomizer
- pressure-swirl atomizer
- air-blast-atomizer
- flat-fan-atomizer
- effervescent-atomizer
- file

For each nonatomizer injection type, you will specify each of the initial conditions listed in *Setting Initial Conditions for the Discrete Phase* (p. 1022), the type of particle that possesses these initial conditions, and any other relevant parameters for the particle type chosen.

You should create a single injection when you want to specify a single value for each of the initial conditions (*Figure 25.5* (p. 1024)). Create a group injection (*Figure 25.6* (p. 1024)) when you want to define a range for one or more of the initial conditions (e.g., a range of diameters or a range of initial positions). To define hollow spray cone injections in 3D problems, create a cone injection (*Figure 25.7* (p. 1025)). To release particles from a surface (either a zone surface or a surface you have defined using the items in the **Surface** menu), you will create a surface injection. (If you create a surface injection, a particle stream will be released from each facet of the surface. You can use the **Bounded** and **Sample Points** options in the **Plane Surface** dialog box to create injections from a rectangular mesh of particles in 3D (see *Plane Surfaces* (p. 1364) for details).

Figure 25.5 Particle Injection Defining a Single Particle Stream

Figure 25.6 Particle Injection Defining an Initial Spatial Distribution of the Particle Streams

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.7 Particle Injection Defining an Initial Spray Distribution of the Particle Velocity

Particle initial conditions (position, velocity, diameter, temperature, and mass flow rate) can also be read from an external file if none of the injection types listed above can be used to describe your injection distribution.

The inputs for setting injections are described in detail in *Defining Injection Properties* (p. 1040).

25.3.2. Particle Types

When you define a set of initial conditions (as described in *Defining Injection Properties* (p. 1040)), you will need to specify the type of particle. The particle types available to you depend on the range of physical models that you have defined.

- A **massless** particle is a discrete element that follows the flow and temperature of the continuous phase. As it has no mass, it has no associated physical properties, and no force is exerted on it. However, you can assign a User-Defined Law to be applied to the massless particle. The massless particle type is available with all ANSYS FLUENT models.
- An **inert** particle is a discrete phase element (particle, droplet, or bubble) that obeys the force balance (Equation 16–1 in the Theory Guide) and is subject to heating or cooling via Law 1 (Inert Heating or Cooling (Law 1/Law 6) in the separate Theory Guide). The inert type is available for all ANSYS FLUENT models.
- A **droplet** particle is a liquid droplet in a continuous-phase gas flow that obeys the force balance (Equation 16–1 in the Theory Guide) and that experiences heating/cooling via Law 1 followed by vaporization and boiling via Laws 2 and 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the Theory Guide). The droplet type is available when heat transfer is being modeled and at least two chemical species are active or the nonpremixed or partially premixed combustion model is active. You should use the ideal gas law to define the gas-phase density (in the **Create/Edit Materials** dialog box, as discussed in *Density Inputs for the Incompressible Ideal Gas Law* (p. 405)) when you select the droplet type.
- A **combusting** particle is a solid particle that obeys the force balance (Equation 16–1 Devolatilization (Law 4)), and a heterogeneous surface reaction via Law 5 (Surface Combustion (Law 5) in the Theory Guide). Finally, the nonvolatile portion of a combusting particle is subject to inert heating via Law 6. You can also include an evaporating material with the combusting particle by selecting the **Wet Combustion** option in the *Set Injection Properties Dialog Box* (p. 2097). This allows you to include a material that evaporates and boils via Laws 2 and 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the Theory Guide) before devolatilization of the particle material begins. The combusting type is available when heat transfer is being modeled and at least three chemical species are active or the nonpremixed combustion model is active. You should use the ideal gas law to define the gas-phase density (in the *Create/Edit Materials Dialog Box* (p. 1736)) when you select the combusting particle type.
- A **multicomponent** particle is, as the name implies, a mixture of droplet particles. These particles contain more than one component, which due to its complexity of assigning a whole particle to one process, have to be modeled by a law that integrates all processes of relevance in one equation. Law 7, the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multicomponent law (Multicomponent Particle Definition (Law 7) in the Theory Guide) is used for such systems. You should use the volume weighted mixing law to define the particle mixture density (in the *Create/Edit Materials Dialog Box* (p. 1736)) when you select the particle-mixture material type.

Particle Type	Description	Laws Activated
Massless	–	–
Inert	inert/heating or cooling	1, 6
Droplet	heating/evaporation/boiling	1, 2, 3, 6
Combusting	heating; evolution of volatiles/swelling; heterogeneous surface reaction	1, 4, 5, 6
Multicomponent	multicomponent droplets/particles	7

25.3.3. Point Properties for Single Injections

For a single injection, you will define the following initial conditions for the particle stream under the **Point Properties** heading (in the *Set Injection Properties Dialog Box* (p. 2097)):

- position

  Set the x, y, and z positions of the injected stream along the Cartesian axes of the problem geometry in the **X-**, **Y-**, and **Z-Position** fields. (**Z-Position** will appear only for 3D problems.)

- velocity

  Set the x, y, and z components of the stream's initial velocity in the **X-**, **Y-**, and **Z-Velocity** fields. (**Z-Velocity** will appear only for 3D problems.)

- diameter

  Set the initial diameter of the injected particle stream in the **Diameter** field.

- temperature

  Set the initial (absolute) temperature of the injected particle stream in the **Temperature** field.

- mass flow rate

  For coupled phase calculations (see *Solution Strategies for the Discrete Phase* (p. 1063)), set the mass of particles per unit time that follows the trajectory defined by the injection in the **Flow Rate** field. Note that in axisymmetric problems the mass flow rate is defined per 2π radians and in 2D problems per unit meter depth (regardless of the reference value for length).

- duration of injection

  For unsteady particle tracking (see *Steady/Transient Treatment of Particles* (p. 1010)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

For the massless particle type, you will only need to define the position of the injection. The particle injection velocity is set by the solver equal to the velocity of the continuous phase at the injection point.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.4. Point Properties for Group Injections

For group injections, you will define the properties position, velocity, diameter, temperature, and flow rate for the **First Point** and **Last Point** in the group. That is, you will define a range of values, ϕ_1 through ϕ_N, for each initial condition ϕ by setting values for ϕ_1 and ϕ_N. ANSYS FLUENT assigns a value of ϕ to the ith injection in the group using a linear variation between the first and last values for ϕ:

$$\phi_i = \phi_1 + \frac{\phi_N - \phi_1}{N-1}(i-1) \qquad \textbf{(25–3)}$$

Thus, for example, if your group consists of 5 particle streams and you define a range for the initial x location from 0.2 to 0.6 meters, the initial x location of each stream is as follows:

- Stream 1: x = 0.2 meters
- Stream 2: x = 0.3 meters
- Stream 3: x = 0.4 meters
- Stream 4: x = 0.5 meters
- Stream 5: x = 0.6 meters

Important

In general, you should supply a range for only one of the initial conditions in a given group—leaving all other conditions fixed while a single condition varies among the stream numbers of the group. Otherwise you may find, for example, that your simultaneous inputs of a spatial distribution and a size distribution have placed the small droplets at the beginning of the spatial range and the large droplets at the end of the spatial range.

The specified flow rate is defined per particle stream and can also be interpolated using *Equation 25–3* (p. 1027). When a Rosin-Rammler sized distribution is specified the total flow rate will be specified.

For the massless particle type, you will only need to define the first and last point of the injection group position. The particle velocities are set by the solver equal to the velocity of the continuous phase at the injection points.

Note that you can use a different method for defining the size distribution of the particles, as discussed below.

25.3.5. Point Properties for Cone Injections

In 3D problems, you can define a hollow or solid cone of particle streams using the **cone** or **solid-cone** injection type, respectively. For both types of cone injections, the inputs are as follows:

- position

  Set the coordinates of the origin of the spray cone in the **X-**, **Y-**, and **Z-Position** fields.

- diameter

  Set the diameter of the particles in the stream in the **Diameter** field.

- temperature

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the temperature of the streams in the **Temperature** field.

- axis

  Set the x, y, and z components of the vector defining the cone's axis in the **X-Axis**, **Y-Axis**, and **Z-Axis** fields.

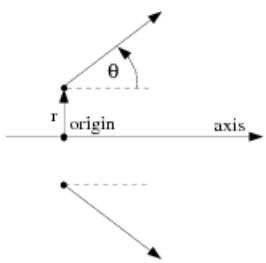
- velocity

  Set the velocity magnitude of the particle streams that will be oriented along the specified spray cone angle in the **Velocity Mag.** field.

- cone angle

  Set the included half-angle, θ, of the hollow spray cone in the **Cone Angle** field, as shown in *Figure 25.8* (p. 1028).

Figure 25.8 Cone Half Angle and Radius



- radius

  A nonzero inner radius can be specified to model injectors that do not emanate from a single point. Set the radius r (defined as shown in *Figure 25.8* (p. 1028)) in the **Radius** field. The particles will be distributed about the axis with the specified radius.

- swirl fraction (hollow cone only)

  Set the fraction of the velocity magnitude to go into the swirling component of the flow in the **Swirl Fraction** field. The direction of the swirl component is defined using the right-hand rule about the axis (a negative value for the swirl fraction can be used to reverse the swirl direction).

- mass flow rate

  For coupled calculations, set the total mass flow rate for the streams in the spray cone in the **Total Flow Rate** field.

The distribution of the particle streams for the solid cone injection is random, as seen in *Figure 25.7* (p. 1025). Furthermore, duplicating this injection may not necessarily result in the same distribution, at the same location.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

For transient calculations, the spatial distribution of streams at the initial injection location is re-calculated at each time step. Sampling different possible trajectories allows a more accurate representation of a solid cone using fewer computational parcels. For steady state calculations, the trajectories are initialized one time and kept the same for subsequent DPM iterations. The trajectories are recalculated when a change in the **Injections** dialog box occurs or when a case and data file are saved. If the residuals and solution change when a small change is made to the injection or when a case and data file are saved, it may mean that there are not enough trajectories being used to represent the solid cone with sufficient accuracy.

Note that you may want to define multiple spray cones emanating from the same initial location in order to specify a size known distribution of the spray or to include a known range of cone angles.

For the massless particle type, you will only need to define position, axis, cone angle and radius. The particle velocities are set by the solver equal to the velocity of the continuous phase at the injection points.

25.3.6. Point Properties for Surface Injections

For surface injections, you will define all the properties described in *Point Properties for Single Injections* (p. 1026) for single injections except for the initial position of the particle streams. The initial positions of the particles will be the location of the data points on the specified surface(s). Note that you will set the **Total Flow Rate** of all particles released from the surface (required for coupled calculations only). If you want, you can scale the individual mass flow rates of the particles by the ratio of the area of the face they are released from to the total area of the surface. To scale the mass flow rates, select the **Scale Flow Rate By Face Area** option under **Point Properties**. For the massless particle type, you will not need to enter any information to define a surface injection. The particle velocities are set by the solver equal to the velocities of the continuous phase at the injection points.

Note that many surfaces have nonuniform distributions of points. If you want to generate a uniform spatial distribution of particle streams released from a surface in 3D, you can create a bounded plane surface with a uniform distribution using the *Plane Surface Dialog Box* (p. 1929), as described in *Plane Surfaces* (p. 1364). In 2D, you can create a rake using the *Line/Rake Surface Dialog Box* (p. 1928), as described in *Line and Rake Surfaces* (p. 1361).

In addition to the option of scaling the flow rate by the face area, the normal direction of a face can be used for the injection direction. To use the face normal direction for the injection direction, select the **Inject Using Face Normal Direction** option under **Point Properties** (*Figure 25.14* (p. 1042)). Once this option is selected, you only need to specify the velocity magnitude of the injection, not the individual components of the velocity magnitude.

Important

Note also that only surface injections from boundary surfaces will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used.

Important

For moving or deforming mesh simulations only zonal surfaces can be selected.

A nonuniform size distribution can be used for surface injections, as described below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.6.1. Using the Rosin-Rammler Diameter Distribution Method

The Rosin-Rammler size distributions described in *Using the Rosin-Rammler Diameter Distribution Method* (p. 1035) for group injections is also available for surface injections. If you select one of the Rosin-Rammler distributions (**rosin-rammler** or **rosin-rammler-logarithmic**), you will need to specify the following parameters under **Point Properties**, in addition to the initial velocity, temperature, and total flow rate:

- **Min. Diameter**

  This is the smallest diameter to be considered in the size distribution.

- **Max. Diameter**

  This is the largest diameter to be considered in the size distribution.

- **Mean Diameter**

  This is the size parameter, $\overline{d}$, in the Rosin-Rammler equation (*Equation 25–5* (p. 1035)).

- **Spread Parameter**

  This is the exponential parameter, n, in *Equation 25–5* (p. 1035).

- **Number of Diameters**

  This is the number of diameters in each distribution (i.e., the number of different diameters in the stream injected from each face of the surface).

ANSYS FLUENT will inject streams of particles from each face on the surface, with diameters defined by the Rosin-Rammler distribution function. The total number of injection streams tracked for the surface injection will be equal to the number of diameters in each distribution (**Number of Diameters**) multiplied by the number of faces on the surface.

25.3.7. Point Properties for Plain-Orifice Atomizer Injections

For a plain-orifice atomizer injection, you will define the following initial conditions under **Point Properties**:

- position

  Set the x, y, and z positions of the injected stream along the Cartesian axes of the problem geometry in the **X-Position**, **Y-Position**, and **Z-Position** fields. (**Z-Position** will appear only for 3D problems).

- axis (3D only)

  Set the x, y, and z components of the vector defining the axis of the orifice in the **X-Axis**, **Y-Axis**, and **Z-Axis** fields.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- mass flow rate

  Set the mass flow rate for the streams in the atomizer in the **Flow Rate** field. Note that in 3D sectors, the flow rate must be appropriate for the sector defined by the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

- duration of injection

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For unsteady particle tracking (see *Steady/Transient Treatment of Particles* (p. 1010)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- vapor pressure

  Set the vapor pressure governing the flow through the internal orifice (p_v in Table 16.2: "List of Governing Parameters for Internal Nozzle Flow" in the Theory Guide) in the **Vapor Pressure** field.

- diameter

  Set the diameter of the orifice in the **Injector Inner Diameter** field (d in Table 16.2: "List of Governing Parameters for Internal Nozzle Flow" in the Theory Guide).

- orifice length

  Set the length of the orifice in the **Orifice Length** field (L in Table 16.2: "List of Governing Parameters for Internal Nozzle Flow" in the Theory Guide).

- radius of curvature

  Set the radius of curvature of the inlet corner in the **Corner Radius of Curvature** field (r in Table 16.2: "List of Governing Parameters for Internal Nozzle Flow" in the Theory Guide).

- nozzle parameter

  Set the constant for the spray angle correlation in the **Constant A** field (C_A in Equation 16–224 in the Theory Guide).

- azimuthal angles

  For 3D sectors, set the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

See The Plain-Orifice Atomizer Model in the Theory Guide for details about how these inputs are used.

25.3.8. Point Properties for Pressure-Swirl Atomizer Injections

For a pressure-swirl atomizer injection, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate, duration of injection (if unsteady), injector inner diameter, and azimuthal angles (if relevant) described in *Point Properties for Plain-Orifice Atomizer Injections* (p. 1030), you will need to specify the following parameters under **Point Properties**:

- spray angle

  Set the value of the spray angle of the injected stream in the **Spray Half Angle** field (θ in Equation 16–233 in the Theory Guide).

- pressure

  Set the absolute pressure upstream of the injection in the **Upstream Pressure** field (p_1 in Table 16.2: "List of Governing Parameters for Internal Nozzle Flow" in the Theory Guide).

- sheet breakup

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Sheet Constant** field ($\ln\left(\frac{\eta_b}{\eta_0}\right)$ in Equation 16–238 in the Theory Guide).

- ligament diameter

  For short waves, set the proportionality constant that linearly relates the ligament diameter, d_L, to the wavelength that breaks up the sheet in the **Ligament Constant** field (see Equation 16–239 – Equation 16–242 in the Theory Guide).

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** for a flat fan atomizer is depicted in *Figure 25.9* (p. 1034).

See The Pressure-Swirl Atomizer Model in the Theory Guide for details about how these inputs are used.

25.3.9. Point Properties for Air-Blast/Air-Assist Atomizer Injections

For an air-blast/air-assist atomizer, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate, duration of injection (if unsteady), injector inner diameter, and azimuthal angles (if relevant) described in *Point Properties for Plain-Orifice Atomizer Injections* (p. 1030), you will need to specify the following parameters under **Point Properties**:

- outer diameter

  Set the outer diameter of the injector in the **Injector Outer Diameter** field. This value is used in conjunction with the **Injector Inner Diameter** to set the thickness of the liquid sheet (t in Equation 16–230 in the Theory Guide).

- spray angle

  Set the initial trajectory of the film as it leaves the end of the orifice in the **Spray Half Angle** field (θ in Equation 16–233 in the Theory Guide).

- relative velocity

  Set the maximum relative velocity that is produced by the sheet and air in the **Relative Velocity** field.

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Sheet Constant** field ($\ln\left(\frac{\eta_b}{\eta_0}\right)$ in Equation 16–238 in the Theory Guide).

- ligament diameter

  For short waves, set the proportionality constant (C_L in Equation 16–241 in the Theory Guide) that linearly relates the ligament diameter, d_L, to the wavelength that breaks up the sheet in the **Ligament Constant** field.

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** for a flat fan atomizer is depicted in *Figure 25.9* (p. 1034).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See The Air-Blast/Air-Assist Atomizer Model in the Theory Guide for details about how these inputs are used.

25.3.10. Point Properties for Flat-Fan Atomizer Injections

The flat-fan atomizer model is available only for 3D models. For this type of injection, you will define the following initial conditions under **Point Properties**:

- arc position

  Set the coordinates of the center point of the arc from which the fan originates in the **X-Center**, **Y-Center**, and **Z-Center** fields (see Figure 16.11: "Flat Fan Viewed from Above and from the Side" in the Theory Guide).

- virtual position

  Set the coordinates of the virtual origin of the fan in the **X-Virtual Origin**, **Y-Virtual Origin**, and **Z-Virtual Origin** fields. This point is the intersection of the lines that mark the sides of the fan (see Figure 16.11: "Flat Fan Viewed from Above and from the Side" in the Theory Guide).

- normal vector

  Set the direction that is normal to the fan in the **X-Fan Normal Vector**, **Y-Fan Normal Vector**, and **Z-Fan Normal Vector** fields.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- mass flow rate

  Set the mass flow rate for the streams in the atomizer in the **Flow Rate** field.

- duration of injection

  For unsteady particle tracking (see *Steady/Transient Treatment of Particles* (p. 1010)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- spray half angle

  Set the initial half angle of the drops as they leave the end of the orifice in the **Spray Half Angle** field.

- orifice width

  Set the width of the orifice (in the normal direction) in the **Orifice Width** field.

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Flat Fan Sheet Constant** field (see Equation 16–238 in the Theory Guide).

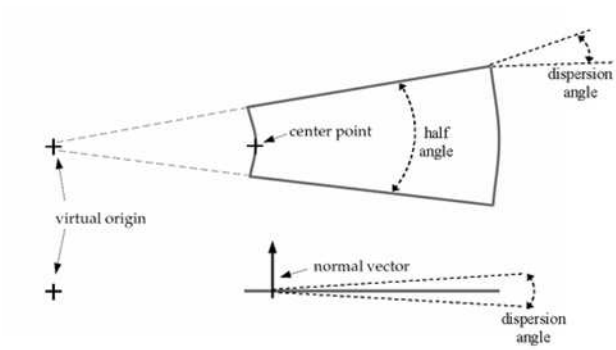
- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** is depicted in *Figure 25.9* (p. 1034).

See The Flat-Fan Atomizer Model in the Theory Guide for details about how these inputs are used.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.9 Flat Fan Viewed from Above and from the Side



25.3.11. Point Properties for Effervescent Atomizer Injections

For an effervescent atomizer injection, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate (including both flashing and nonflashing components), duration of injection (if unsteady), vapor pressure, injector inner diameter, and azimuthal angles (if relevant) described in *Point Properties for Plain-Orifice Atomizer Injections* (p. 1030), you will need to specify the following parameters under **Point Properties**:

- mixture quality

  Set the mass fraction of the injected mixture that vaporizes in the **Mixture Quality** field (x in Equation 16–248 in the Theory Guide).

- saturation temperature

  Set the saturation temperature of the volatile substance in the **Saturation Temp.** field.

- droplet dispersion

  Set the parameter that controls the spatial dispersion of the droplet sizes in the **Dispersion Constant** field (C_{eff} in Equation 16–248 in the Theory Guide).

- spray angle

  Set the initial trajectory of the film as it leaves the end of the orifice in the **Maximum Half Angle** field.

See The Effervescent Atomizer Model in the Theory Guide for details about how these inputs are used.

25.3.12. Point Properties for File Injections

The file for a file injection has the following form:

```
(( x y z u v w diameter temperature mass-flow) name )
```

with all of the parameters in SI units. All the parentheses are required, but the `name` is optional.

Sample files generated during sampling of trajectories for steady particles (see *Sampling of Trajectories* (p. 1082)) can also be used as injection files since they have a similar file format.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.13. Using the Rosin-Rammler Diameter Distribution Method

For liquid sprays, a convenient representation of the droplet size distribution is the Rosin-Rammler expression. The complete range of sizes is divided into an adequate number of discrete intervals; each represented by a mean diameter for which trajectory calculations are performed. If the size distribution is of the Rosin-Rammler type, the mass fraction of droplets of diameter greater than d is given by

$$Y_d = e^{-(d/\bar{d})^n} \tag{25–4}$$

where $\bar{d}$ is the size constant and n is the size distribution parameter.

By default, you will define the size distribution of particles by inputting a diameter for the first and last points and using the linear equation (*Equation 25–3* (p. 1027)) to vary the diameter of each particle stream in the group. When you want a different mass flow rate for each particle/droplet size, however, the linear variation may not yield the distribution you need. Your particle size distribution may be defined most easily by fitting the size distribution data to the Rosin-Rammler equation. In this approach, the complete range of particle sizes is divided into a set of discrete size ranges, each to be defined by a single stream that is part of the group. Assume, for example, that the particle size data obeys the following distribution:

Diameter Range (μm)	**Mass Fraction in Range**
0–70	0.05
70–100	0.10
100–120	0.35
120–150	0.30
150–180	0.15
180–200	0.05

The Rosin-Rammler distribution function is based on the assumption that an exponential relationship exists between the droplet diameter, d, and the mass fraction of droplets with diameter greater than d, Y_d:

$$Y_d = e^{-(d/\bar{d})^n} \tag{25–5}$$

ANSYS FLUENT refers to the quantity $\bar{d}$ in *Equation 25–5* (p. 1035) as the **Mean Diameter** and to n as the **Spread Parameter**. These parameters are input by you (in the *Set Injection Properties Dialog Box* (p. 2097) under the **First Point** heading) to define the Rosin-Rammler size distribution. To solve for these parameters, you must fit your particle size data to the Rosin-Rammler exponential equation. To determine these inputs, first recast the given droplet size data in terms of the Rosin-Rammler format. For the example data provided above, this yields the following pairs of d and Y_d:

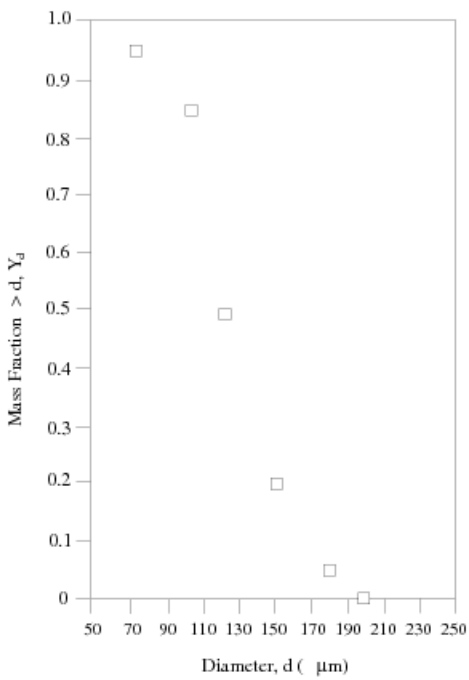
Diameter, d (μm)	**Mass Fraction with Diameter Greater than d, Y_d**
70	0.95
100	0.85
120	0.50
150	0.20

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diameter, d (μm)	Mass Fraction with Diameter Greater than d, Y_d
180	0.05
200	(0.00)

A plot of Y_d vs. d is shown in *Figure 25.10* (p. 1036).

Figure 25.10 Example of Cumulative Size Distribution of Particles



Next, derive values of $\overline{d}$ and n such that the data in *Figure 25.10* (p. 1036) fit *Equation 25–5* (p. 1035). The value for $\overline{d}$ is obtained by noting that this is the value of d at which $Y_d = e^{-1} \approx 0.368$. From *Figure 25.10* (p. 1036), you can estimate that this occurs for $d \approx 131\,\mu$m. The numerical value for n is given by

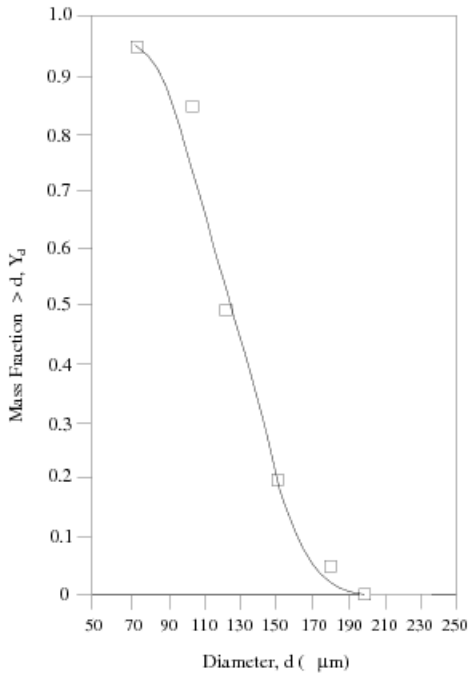
$$n = \frac{\ln\left(-\ln Y_d\right)}{\ln\left(d/\overline{d}\right)} \qquad \textbf{(25–6)}$$

By substituting the given data pairs for Y_d and $d/\overline{d}$ into this equation, you can obtain values for n and find an average. Doing so yields an average value of n = 4.52 for the example data above. The resulting Rosin-Rammler curve fit is compared to the example data in *Figure 25.11* (p. 1037). You can input values for $\overline{d}$ and n, as well as the diameter range of the data and the total mass flow rate for the combined individual size ranges, using the *Set Injection Properties Dialog Box* (p. 2097).

This technique of fitting the Rosin-Rammler curve to spray data is used when reporting the Rosin-Rammler diameter and spread parameter in the **Discrete Phase Summary** dialog box in *Summary Reporting of Current Particles* (p. 1085).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.11 Rosin-Rammler Curve Fit for the Example Particle Size Data



A second Rosin-Rammler distribution is also available based on the natural logarithm of the particle diameter. If in your case, the smaller-diameter particles in a Rosin-Rammler distribution have higher mass flows in comparison with the larger-diameter particles, you may want better resolution of the smaller-diameter particle streams, or "bins". You can therefore choose to have the diameter increments in the Rosin-Rammler distribution done uniformly by $\ln d$.

In the standard Rosin-Rammler distribution, a particle injection may have a diameter range of 1 to 200 μm. In the logarithmic Rosin-Rammler distribution, the same diameter range would be converted to a range of $\ln 1$ to $\ln 200$, or about 0 to 5.3. In this way, the mass flow in one bin would be less-heavily skewed as compared to the other bins.

When a Rosin-Rammler size distribution is being defined for the group of streams, you should define (in addition to the initial velocity, position, and temperature) the following parameters, which appear under the heading for the **First Point**:

- **Total Flow Rate**

  This is the total mass flow rate of the N streams in the group. Note that in axisymmetric problems this mass flow rate is defined per 2π radians and in 2D problems per unit meter depth.

- **Min. Diameter**

  This is the smallest diameter to be considered in the size distribution.

- **Max. Diameter**

  This is the largest diameter to be considered in the size distribution.

- **Mean Diameter**

  This is the size parameter, $\overline{d}$, in the Rosin-Rammler equation (*Equation 25–5* (p. 1035)).

- **Spread Parameter**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is the exponential parameter, n, in *Equation 25–5* (p. 1035).

25.3.13.1. The Stochastic Rosin-Rammler Diameter Distribution Method

For atomizer injections, a Rosin-Rammler distribution is assumed for the particles exiting the injector. In order to decrease the number of particles necessary to accurately describe the distribution, the diameter distribution function is randomly sampled for each instance where new particles are introduced into the domain.

The Rosin-Rammler distribution can be written as

$$1 - Y = \exp\left[-\left(\frac{D}{\bar{d}}\right)^n \right] \tag{25–7}$$

where Y is the mass fraction smaller than a given diameter D, $\bar{d}$ is the Rosin-Rammler diameter and n is the Rosin-Rammler exponent. This expression can be inverted by taking logs of both sides and rearranging,
where Y is the mass fraction smaller than a given diameter D, $\bar{d}$ is the Rosin-Rammler diameter and n is the Rosin-Rammler exponent. This expression can be inverted by taking logs of both sides and rearranging,

$$D = \bar{d}\ (-\ln(1 - Y))^{1/n}. \tag{25–8}$$

Given a mass fraction Y along with parameters $\bar{d}$ and n, this function will explicitly provide a diameter, D. Diameters for the atomizer injectors described in *Point Properties for Plain-Orifice Atomizer Injections* (p. 1030) are obtained by uniformly sampling Y in *Equation 25–8* (p. 1038).

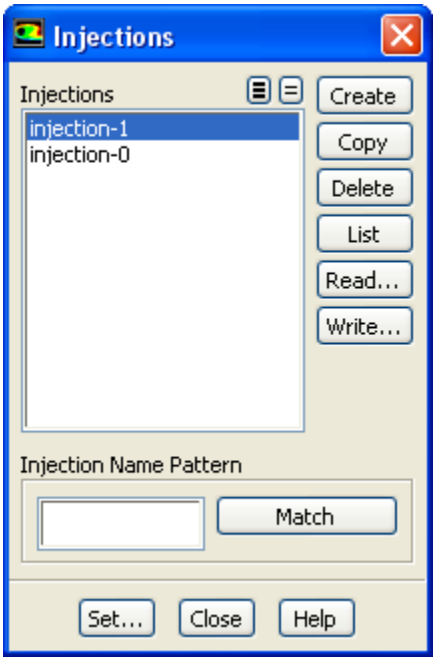
25.3.14. Creating and Modifying Injections

You will use the *Injections Dialog Box* (p. 2096) (*Figure 25.12* (p. 1039)) to create, copy, delete, list, read, and write injections.

Define $\rightarrow$ **Injections...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.12 The Injections Dialog Box



(You can also click the **Injections...** button in the *Discrete Phase Model Dialog Box* (p. 1723) to open the **Injections** dialog box.)

25.3.14.1. Creating Injections

To create an injection, click the **Create** button. The *Set Injection Properties Dialog Box* (p. 2097) will open automatically to allow you to set the injection properties (as described in *Defining Injection Properties* (p. 1040)). After the injection is created, the new injection will appear in the **Injections** list, in the **Injections** dialog box.

25.3.14.2. Modifying Injections

To modify an existing injection, select its name in the **Injections** list and click the **Set...** button. The **Set Injection Properties** dialog box will open, and you can modify the properties as needed.

If you have two or more injections for which you want to set some of the same properties, select their names in the **Injections** list and click the **Set...** button. The **Set Multiple Injection Properties** dialog box will open, which will allow you to set the common properties. For instructions about using this dialog box, see *Defining Properties Common to More than One Injection* (p. 1047).

25.3.14.3. Copying Injections

To copy an existing injection to a new injection, select the existing injection in the **Injections** list and click the **Copy** button. The *Set Injection Properties Dialog Box* (p. 2097) will open with a new injection that has the same properties as the injection you selected. This is useful if you want to set another injection with similar properties.

25.3.14.4. Deleting Injections

You can delete an injection by selecting its name in the **Injections** list and clicking the **Delete** button.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.14.5. Listing Injections

To list the initial conditions for the particle streams in the selected injection, click the **List** button. ANSYS FLUENT reports the initial conditions (in SI units) in the console under various columns:

- The particle stream number is in the column headed `NO`.
- The particle type (`IN`for inert, `DR`for droplet, or `CP` for combusting particle) is in the column headed `TYP`.
- The x, y, and z positions are in the columns headed `(X)`, `(Y)`, and `(Z)`.
- The x, y, and z velocities are in the columns headed `(U)`, `(V)`, and `(W)`.
- The temperature is in the column headed `(T)`.
- The diameter is in the column headed `(DIAM)`.
- The mass flow rate in the column headed `(MFLOW)`.

25.3.14.6. Reading and Writing Injections

To transfer information about DPM injections from one case file to another, use the **Read...** and **Write...** buttons. You can write selected injections to a file, which can be read into a different ANSYS FLUENT session, simplifying the setup of new case files. To write the injection, select the injection from the list, then click **Write...**. *The **Select File** Dialog Box* (p. 33) will open where you can enter the name of your injection file. To read in an injection, click **Read...** to open *The **Select File** Dialog Box* (p. 33) where you will select the injection file to read in. If the injection that you imported has the same name as that in your current case, then ANSYS FLUENT will rename the imported injection.

After reading injections, you may need to visit the **Injections** dialog box to modify the settings for the injection material and the DPM laws since the presumed settings may have changed in the current case file setup.

25.3.14.7. Shortcuts for Selecting Injections

ANSYS FLUENT provides a shortcut for selecting injections with names that match a specified pattern. To use this shortcut, enter the pattern under **Injection Name Pattern** and then click **Match** to select the injections with names that match the specified pattern. For example, if you specify `drop*`, all injections that have names beginning with **drop** (e.g., **drop-1**, **droplet**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `drop?`, all surfaces with names consisting of **drop** followed by a single character will be selected (or deselected, if they are all selected already).

25.3.15. Defining Injection Properties

Once you have created an injection (using the *Injections Dialog Box* (p. 2096), as described in *Creating and Modifying Injections* (p. 1038)), you will use the *Set Injection Properties Dialog Box* (p. 2097) (*Figure 25.13* (p. 1041)) to define the injection properties. (Remember that this dialog box will open when you create a new injection, or when you select an existing injection and click the **Set...** button in the **Injections** dialog box.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.13 The Set Injection Properties Dialog Box

The procedure for defining an injection is as follows:

1. If you want to change the name of the injection from its default name, enter a new one in the **Injection Name** field. This is recommended if you are defining a large number of injections so you can easily distinguish them. When assigning names to your injections, keep in mind the selection shortcut described in *Creating and Modifying Injections* (p. 1038).

2. Choose the type of injection in the **Injection Type** drop-down list. The eleven choices (**single**, **group**, **cone**, **solid-cone**, **surface**, **plain-orifice-atomizer**, **pressure-swirl-atomizer**, **air-blast-atomizer**, **flat-fan-atomizer**, **effervescent-atomizer**, and **file**) are described in *Injection Types* (p. 1024). Note that if you select any of the atomizer models, you will also need to set the **Viscosity** and **Droplet Surface Tension** in the **Create/Edit Materials** dialog box.

   Important

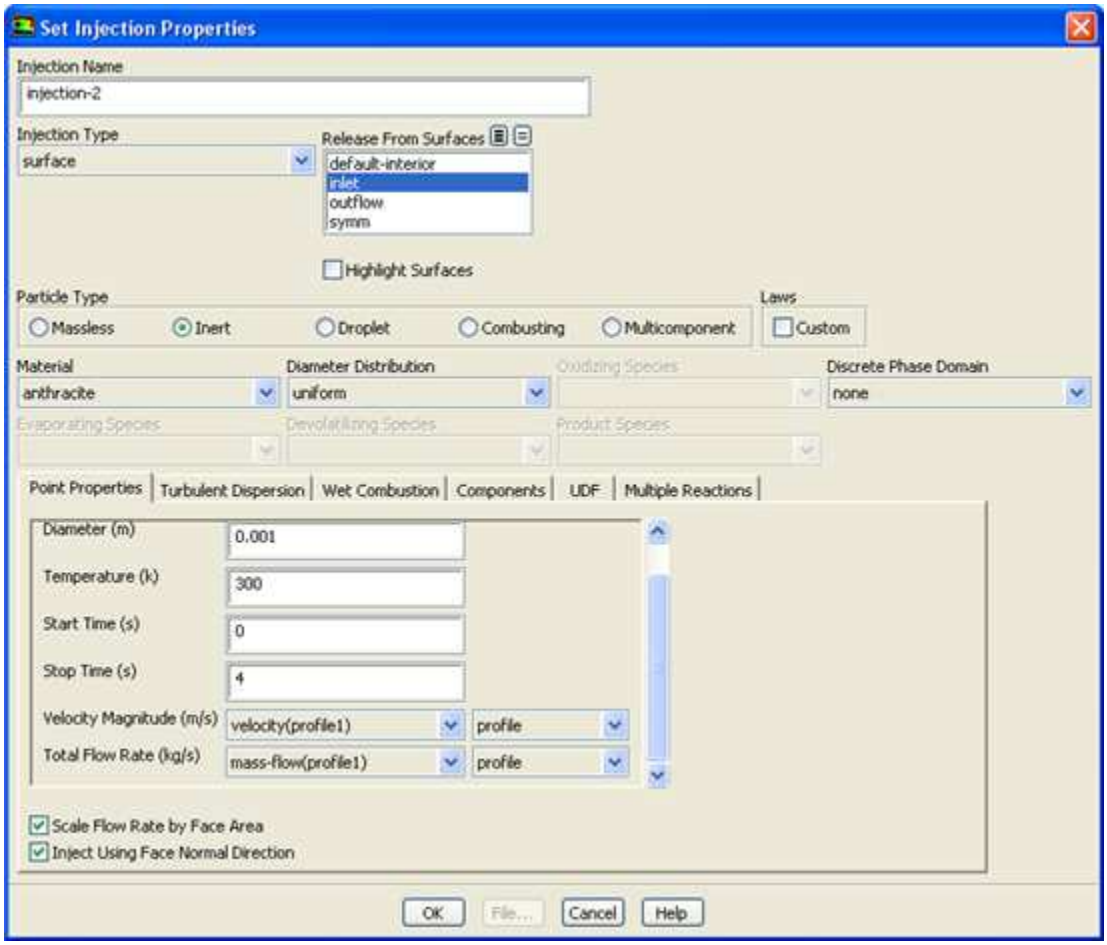
   Note that only surface injections from boundary surfaces will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used.

3. If you are defining a **single** injection, go to the next step. For a **group**, **cone**, **solid-cone**, or any of the **atomizer** injections, set the **Number of Particle Streams** in the group, spray cone, or atomizer.

   If you are defining a **surface** injection (see *Figure 25.14* (p. 1042)), choose the surface(s) from which the particles will be released in the **Release From Surfaces** list. If you are reading the injection from a file, click the **File...** button at the bottom of the *Set Injection Properties Dialog Box* (p. 2097) and specify the file to be read in the resulting **Select File** dialog box. The parameters in the injection file must be in SI units.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.14 Setting Surface Injection Properties



4. Select **Massless**, **Inert**, **Droplet**, **Combusting**, or **Multicomponent** as the **Particle Type**. The available types are described in *Particle Types* (p. 1025).

5. Choose the material

   for the particle(s) in the **Material** drop-down list. If this is the first time you have created a particle of this type, you can choose from all of the materials of this type defined in the database. If you have already created a particle of this type, the only available material will be the material you selected for that particle. You can define additional materials by copying them from the database or creating them from scratch, as discussed in *Setting Discrete-Phase Physical Properties* (p. 1057) and described in detail in *Using the Materials Task Page* (p. 387).

   Important

   Note that you will not choose a **Material** for a **Massless** particle type.

6. If you are defining a **group** or **surface** injection and you want to change from the default **linear** (for group injections) or **uniform** (for surface injections) interpolation method used to determine the size of the particles, select **rosin-rammler** or **rosin-rammler-logarithmic** in the **Diameter Distribution** drop-down list. The Rosin-Rammler method for determining the range of diameters for a group injection is described in *Using the Rosin-Rammler Diameter Distribution Method* (p. 1035).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If you have created a customized particle law using user-defined functions, enable the **Custom** option under **Laws** and specify the appropriate laws as described in *Custom Particle Laws* (p. 1046).

8. If your particle type is **Inert**, go to the next step. If you are defining **Droplet** particles, select the gas phase species created by the vaporization and boiling laws (Laws 2 and 3) in the **Evaporating Species** drop-down list.

   If you are defining **Combusting** particles, select the gas phase species created by the devolatilization law (Law 4) in the **Devolatilizing Species** drop-down list, the gas phase species that participates in the surface char combustion reaction (Law 5) in the **Oxidizing Species** list, and the gas phase species created by the surface char combustion reaction (Law 5) in the **Product Species** list. Note that if the **Combustion Model** for the selected combusting particle material (in the **Create/Edit Materials** dialog box) is the **multiple-surface-reaction** model, then the **Oxidizing Species** and **Product Species** lists will be disabled because the reaction stoichiometry has been defined in the mixture material.

   If you are defining **Multicomponent** particles, law 7 will go into effect. Notice that the **Components** tab will become active when this particle type is selected. See below for information on the **Components** tab.

9. Click the **Point Properties** tab (the default), and specify the point properties (position, velocity, diameter, temperature, and—if appropriate—mass flow rate and any atomizer-related parameters) as described for each injection type in *Point Properties for Single Injections* (p. 1026) – *Point Properties for Effervescent Atomizer Injections* (p. 1034).

   For surface injections, you can enable the **Scale Flow Rate by Face Area** and you can choose the injection direction. To use the face normal direction for the injection direction, select the **Inject Using Face Normal Direction** option under **Point Properties** (*Figure 25.14* (p. 1042)). Once this option is selected, you only need to specify the velocity magnitude of the injection, not the individual components of the velocity magnitude.

10. If the flow is turbulent and you wish to include the effects of turbulence on the particle dispersion, click the **Turbulent Dispersion** tab, enable the **Discrete Random Walk Model** under **Stochastic tracking** or the **Cloud Model**, and set the related parameters as described in *Specifying Turbulent Dispersion of Particles* (p. 1044).

11. If your combusting particle includes an evaporating material, click the **Wet Combustion** tab, select the **Wet Combustion Model** option, and then select the material that is evaporating/boiling from the particle before devolatilization begins in the **Liquid Material** drop-down list. You should also set the volume fraction of the liquid present in the particle by entering the value of the **Liquid Fraction**. Finally, select the gas phase species created by the evaporating and boiling laws in the **Evaporating Species** drop-down list in the top part of the dialog box.

12. If you include multicomponent droplets as the material in your discrete phase model, the **Components** tab will become active. In this tab, you will specify the **Mass Fraction** of each of the components. Note that the sum of the Mass fractions should add up to unity, otherwise ANSYS FLUENT will adjust the values such that you have a sum of 1 for the mass fraction, and will prompt you to accept the entry. Under **Evaporating Species**, select **not-vaporizing** if the component in the particle does not vaporize. Otherwise, select the species that will be vaporized.

    To change the components of a materials from the **FLUENT Database Materials** dialog box, or define the droplet materials in the **Create/Edit Materials** dialog box, then add them to the **Selected Species** list in the **Species** dialog box by clicking the **Edit...** button (in the **Create/Edit Materials** dialog box) next to **Mixture Species**.

13. If you want to use a user-defined function to initialize the injection properties, click the **UDF** tab to access the UDF inputs. You can select an **Initialization** function under **User-Defined Functions** to modify injection properties at the time the particles are injected into the domain. This allows the pos-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ition and/or properties of the injection to be set as a function of flow conditions. More information about user-defined functions can be found in the UDF Manual.

14. If you have defined more than one particle surface species, for example, carbon (**C<s>**) and sulfur (**S<s>**), you will need to specify the mass fraction of each particle surface specie in the combusting particle. To do so, click the **Multiple Reactions** tab, and enter the **Species Mass Fractions**. These mass fractions refer to the combustible fraction of the combusting particle, and should sum to 1. If there is only one surface specie in the mixture material, the mass fraction of that specie will be set to 1, and you will not specify anything under **Multiple Surface Reactions**.

25.3.16. Specifying Turbulent Dispersion of Particles

As mentioned in *Defining Injection Properties* (p. 1040), you can choose for each injection stochastic tracking or cloud tracking as the method for modeling turbulent dispersion of particles.

25.3.16.1. Stochastic Tracking

For turbulent flows, if you choose to use the stochastic tracking technique, you must enable the **Discrete Random Walk Model** and specify the **Number of Tries**. Stochastic tracking includes the effect of turbulent velocity fluctuations on the particle trajectories using the DRW model described in Stochastic Tracking the Theory Guide.

1. Click the **Turbulent Dispersion** tab in the **Set Injection Properties** dialog box.
2. Enable stochastic tracking by turning on the **Discrete Random Walk Model** under **Stochastic Tracking**.
3. Specify the **Number of Tries**:

   An input of 1 or greater tells ANSYS FLUENT to include turbulent velocity fluctuations in the particle force balance as in Equation 16–14 in the Theory Guide. The trajectory is computed more than once if your input exceeds 1: two trajectory calculations are performed if you input 2, three trajectory calculations are performed if you input 3, etc. Each trajectory calculation includes a new stochastic representation of the turbulent contributions to the trajectory equation.

   When a sufficient number of tries is requested, the trajectories computed will include a statistical representation of the spread of the particle stream due to turbulence. Note that for unsteady particle tracking, the **Number of Tries** is set to 1 if using stochastic tracking.

If you want the characteristic lifetime of the eddy to be random (Equation 16–25 in the Theory Guide), enable the **Random Eddy Lifetime** option. You will generally not need to change the **Time Scale Constant** (C_L in Equation 16–16 in the Theory Guide) from its default value of 0.15, unless you are using the Reynolds Stress turbulence model (RSM), in which case a value of 0.3 is recommended.

Figure 25.15 (p. 1045) illustrates a discrete phase trajectory calculation computed via the "mean" tracking (number of tries = 0) and *Figure 25.16* (p. 1045) illustrates the "stochastic" tracking (number of tries > 1) option.

When multiple stochastic trajectory calculations are performed, the momentum and mass defined for the injection are divided evenly among the multiple particle/droplet tracks, and are thus spread out in terms of the interphase momentum, heat, and mass transfer calculations. Including turbulent dispersion in your model can thus have a significant impact on the effect of the particles on the continuous phase when coupled calculations are performed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.15 Mean Trajectory in a Turbulent Flow

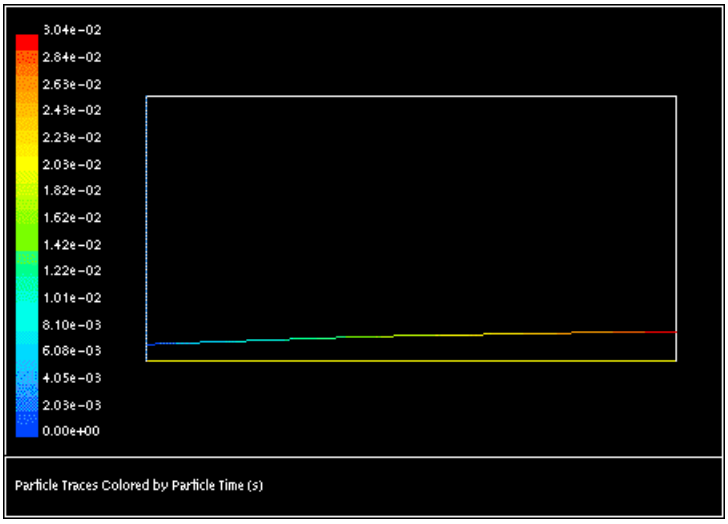
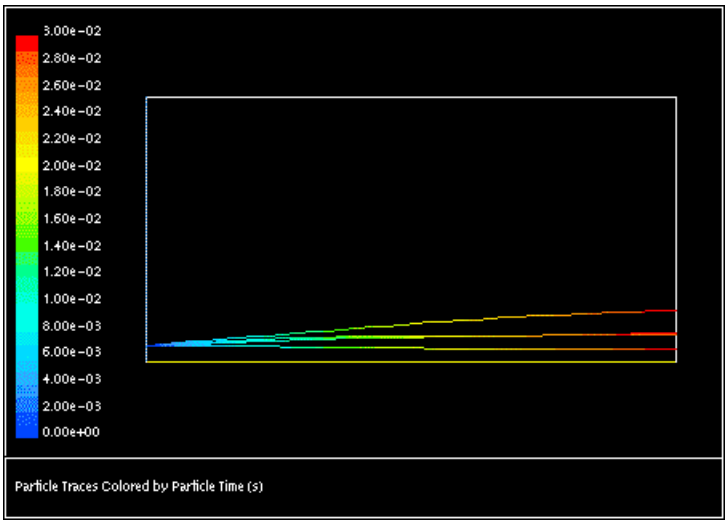


Figure 25.16 Stochastic Trajectories in a Turbulent Flow



25.3.16.2. Cloud Tracking

For turbulent flows, you can also include the effects of turbulent dispersion on the injection. Note that cloud tracking is not available for the massless particle type. When cloud tracking is used, the trajectory will be tracked as a cloud of particles about a mean trajectory, as described in Particle Cloud Tracking in the Theory Guide.

1. Click the **Turbulent Dispersion**tab in the **Set Injection Properties** dialog box.
2. Enable cloud tracking by turning on the **Cloud Model** under **Cloud Tracking**.
3. Specify the minimum and maximum cloud diameters. Particles enter the domain with an initial cloud diameter equal to the **Min. Cloud Diameter**. The particle cloud's maximum allowed diameter is specified by the **Max. Cloud Diameter**.

   You may want to restrict the **Max. Cloud Diameter** to a relevant length scale for the problem to improve computational efficiency in complex domains where the mean trajectory may become stuck in recirculation regions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

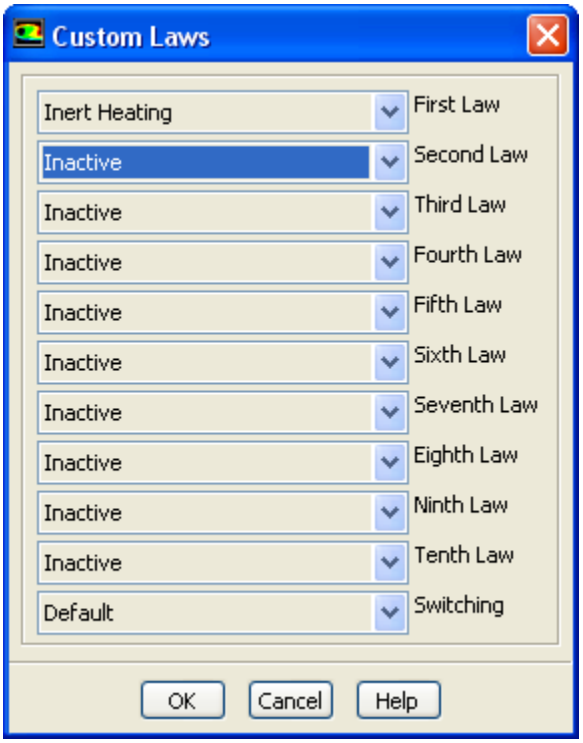
Note that this model is available only in serial or for shared memory tracking.

25.3.17. Custom Particle Laws

If the standard ANSYS FLUENT laws, Laws 1 through 7, do not adequately describe the physics of your discrete phase model, you can modify them by creating custom laws with user-defined functions. More information about user-defined functions can be found in the UDF Manual. You can also create custom laws by using a subset of the existing ANSYS FLUENT laws (e.g., Laws 1, 2, and 4), or a combination of existing laws and user-defined functions.

Once you have defined and loaded your user-defined function(s), you can create a custom law by enabling the **Custom** option under **Laws** in the *Set Injection Properties Dialog Box* (p. 2097). This will open the *Custom Laws Dialog Box* (p. 2102). In the drop-down list to the left of each of the six particle laws, you can select the appropriate particle law for your custom law. Each list contains the available options that can be chosen (the standard laws plus any user-defined functions you have loaded).

Figure 25.17 The Custom Laws Dialog Box



There is a seventh drop-down list in the *Custom Laws Dialog Box* (p. 2102) labeled **Switching**. You may wish to have ANSYS FLUENT vary the laws used depending on conditions in the model. You can customize the way ANSYS FLUENT switches between laws by selecting a user-defined function from this drop-down list.

An example of when you might want to use a custom law might be to replace the standard devolatilization law with a specialized devolatilization law that more accurately describes some unique aspects of your model. After creating and loading a user-defined function that details the physics of your devolatilization law, you would visit the *Custom Laws Dialog Box* (p. 2102) and replace the standard devolatilization law (Law 2) with your user-defined function.

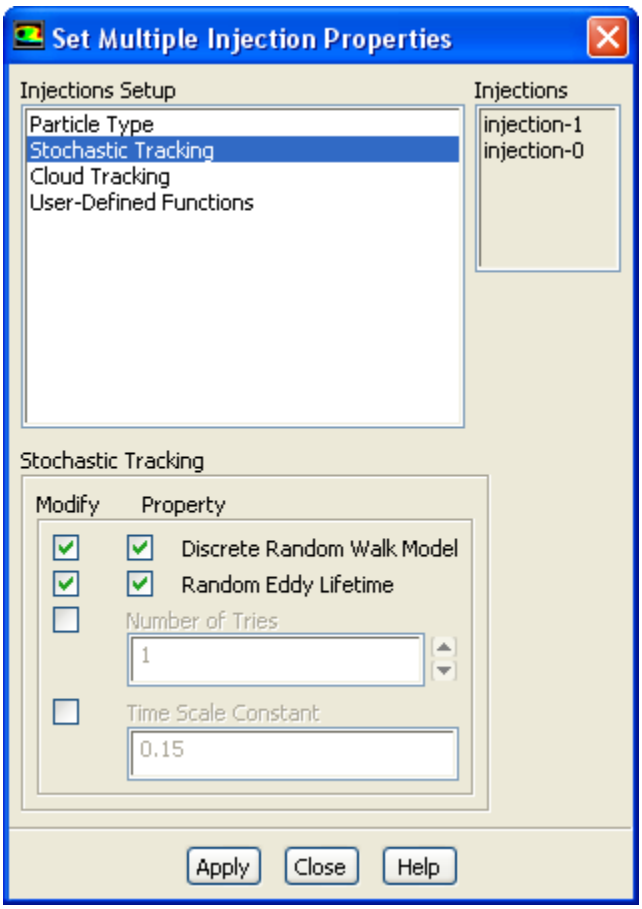
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3.18. Defining Properties Common to More than One Injection

If you have a number of injections for which you want to set the same properties, ANSYS FLUENT provides a shortcut so that you do not need to visit the **Set Injection Properties** dialog box for each injection to make the same changes.

As described in *Defining Injection Properties* (p. 1040), if you select more than one injection in the **Injections** dialog box, clicking the **Set...** button will open the **Set Multiple Injection Properties** dialog box (*Figure 25.18* (p. 1047)) instead of the **Set Injection Properties** dialog box.

Figure 25.18 The Set Multiple Injection Properties Dialog Box



Depending on the type of injections you have selected (single, group, atomizers, etc.), there will be different categories of properties listed under **Injections Setup**. The names of these categories correspond to the headings within the **Set Injection Properties** dialog box (e.g., **Particle Type** and **Stochastic Tracking**). Only those categories that are appropriate for all of your selected injections (which are shown in the **Injections** list) will be listed. If all of these injections are of the same type, more categories of properties will be available for you to modify. If the injections are of different types, you will have fewer categories to select from.

25.3.18.1. Modifying Properties

To modify a property, perform the following steps:

1. Select the appropriate category in the **Injections Setup** list. For example, if you want to set the same flow rate for all of the selected injections, select **Point Properties**. The dialog box will expand to show the properties that appear under that heading in the **Set Injection Properties** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the property (or properties) to be modified, as described below.
3. Click **Apply**. ANSYS FLUENT will report the change in the console window.

Important

You must click **Apply** to save the property settings within each category. If, for example, you want to modify the flow rate and the stochastic tracking parameters, you will need to select **Point Properties** in the **Injections Setup** list, specify the flow rate, and click **Apply**. You would then repeat the process for the stochastic tracking parameters, clicking **Apply** again when you are done.

There are two types of properties that can be modified using the **Set Multiple Injection Properties** dialog box.

The first type involves one of the following actions:

- selecting a value from a drop-down list
- choosing an option using a radio button

The second type involves one of the following actions:

- entering a value in a field
- turning an option on or off

Setting the first type of property works the same way as in the **Set Injection Properties** dialog box. For example, if you select **Particle Type** in the **Injections Setup** list, the dialog box will expand to show the portion of the **Set Injection Properties** dialog box where you choose the particle type. You can simply choose the desired type and click **Apply**.

Setting the second type of property requires an additional step. If you select a category in the **Injections Setup** list that contains this type of property, the expanded portion of the dialog box will look like the corresponding part of the **Set Injection Properties** dialog box, with the addition of **Modify** check buttons (see *Figure 25.18* (p. 1047)). To change one of the properties, first turn on the **Modify** check button to its left, and then specify the desired status or value.

For example, if you would like to enable stochastic tracking, first turn on the **Modify** check button to the left of **Stochastic Model**. This will make the property active so you can modify its status. Then, under **Property**, turn on the **Stochastic Model** check button. (Be sure to click **Apply** when you are done setting stochastic tracking parameters.)

If you would like to change the value of **Number of Tries**, select the **Modify** check button to its left to make it active, and then enter the new value in the field. Make sure you click **Apply** when you have finished modifying the stochastic tracking properties.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The setting for a property that has not been activated with the **Modify** check button is not relevant, because it will not be applied to the selected injections when you click **Apply**. After you turn on **Modify** for a particular property, clicking **Apply** will modify that property for all of the selected injections, so make sure that you have the settings the way that you want them before you do this. If you make a mistake, you will have to return to the **Set Injection Properties** dialog box for each injection to fix the incorrect setting, if it is not possible to do so in the **Set Multiple Injection Properties** dialog box.

25.3.18.2. Modifying Properties Common to a Subset of Selected Injections

Note that it is possible to change a property that is relevant for only a subset of the selected injections. For example, if some of the selected injections are using stochastic tracking and some are not, enabling the **Random Eddy Lifetime** option and clicking **Apply** will turn this option on only for those injections that are using stochastic tracking. The other injections will be unaffected.

25.3.19. Point Properties for Transient Injections

Simulations of transient particles often require time dependent injection conditions. Potentially most of the point properties may change over time. One method to accomplish this is the use of an **Initialization** function as described in the UDF manual.

Define → **User-Defined** → **Function Hooks...**

An easier way is to use transient profiles containing one or more variables based on the time or crank angle that can be assigned to various point properties: (Total) **Mass Flow Rate**, **X-, Y-, Z- Velocity**, and **Velocity Magnitude** depending on the injection type selected. See *Point Properties for Single Injections* (p. 1026) to *Point Properties for Effervescent Atomizer Injections* (p. 1034) for the description of the individual properties.

Before transient profile variables can be assigned to point properties of injections, a profile files has to be read into ANSYS FLUENT using the

File → **Read** → **Profile...**

menu item. In the **Select File** dialog box a **transient profile** in tabular format with extension `.ttab` has to be chosen. Alternatively, you can read this file into ANSYS FLUENT using the `read-transient-table` text command.

`file` → `read-transient-table`

The profile name should not exceed 63 characters. See *Defining Transient Cell Zone and Boundary Conditions* (p. 376) for the format description. The following example illustrates an injection within 3 intervals of 2.5 milliseconds injection time, where the second injection has an elevated velocity.

```
mv-profile 3 13 0
time          mass-flow     velocity
0              0              0.1
0.00999       0              0.1
0.01           0.001         0.1
0.0125         0.001         0.1
0.01251       0              0.1
0.01999       0              0.1
0.02           0.001         5
0.0225         0.001         5
0.02251       0              0.1
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.02999       0              0.1
0.03           0.001         0.1
0.0325        0.001         0.1
0.03251       0              0.1
```

For the setting of point properties go to the **Point Properties** tab in the **Set Injection Properties** dialog box.

When transient profiles are loaded, you can choose either a **constant** method (the default) or **profile** from the **Method** drop-down list. If you select **profile**, you will also need to choose a profile from the corresponding drop-down list that appears in the **Value** column. The variable angle or time from the file is used as an independent interpolation variable and cannot be chosen as a profile.

25.4. Setting Boundary Conditions for the Discrete Phase

When a particle reaches a physical boundary (e.g., a wall or inlet boundary) in your model, ANSYS FLUENT applies a discrete phase boundary condition to determine the fate of the trajectory at that boundary. One of several contingencies may arise:

- The particle may be reflected via an elastic or inelastic collision.
- The particle may escape through the boundary. The particle is lost from the calculation at the point where it impacts the boundary.
- The particle may be trapped at the wall. Nonvolatile material is lost from the calculation at the point of impact with the boundary; volatile material present in the particle or droplet is released to the vapor phase at this point.
- The particle may pass through an internal boundary zone, such as radiator or porous jump.
- The particle may slide along the wall, depending on particle properties and impact angle.
- The particle may form a film (Wall-Film Model).

You also have the option of implementing a user-defined function to model the particle behavior when hitting the boundary. More information about user-defined functions can be found in the UDF Manual.

The boundary condition, or trajectory fate, can be defined separately for each zone in your ANSYS FLUENT model.

For additional information, please see the following sections:

25.4.1. Discrete Phase Boundary Condition Types
25.4.2. Setting Particle Erosion and Accretion Parameters

25.4.1. Discrete Phase Boundary Condition Types

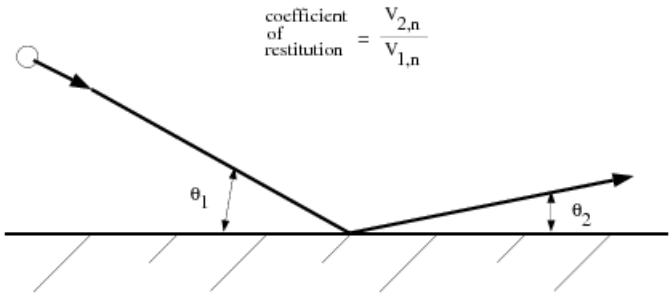
The available boundary conditions are

- **reflect**

  The particle rebounds the off the boundary in question with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051).)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.19 "Reflect" Boundary Condition for the Discrete Phase



The normal coefficient of restitution defines the amount of momentum in the direction normal to the wall that is retained by the particle after the collision with the boundary [*89*] (p. 2213):

$$e_n = \frac{v_{2,n}}{v_{1,n}} \tag{25–9}$$

where v_n is the particle velocity normal to the wall and the subscripts 1 and 2 refer to before and after collision, respectively. Similarly, the tangential coefficient of restitution, e_t, defines the amount of momentum in the direction tangential to the wall that is retained by the particle.

A normal or tangential coefficient of restitution equal to 1.0 implies that the particle retains all of its normal or tangential momentum after the rebound (an elastic collision). A normal or tangential coefficient of restitution equal to 0.0 implies that the particle retains none of its normal or tangential momentum after the rebound.

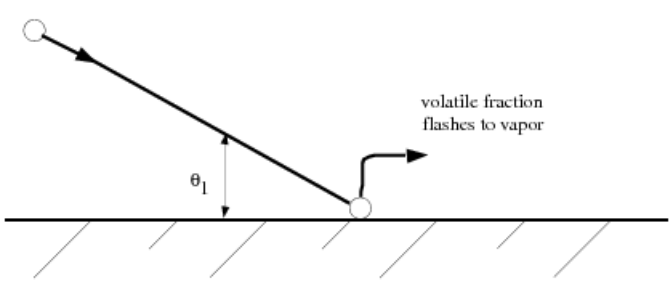
Nonconstant coefficients of restitution can be specified for wall zones with the **reflect** type boundary condition. The coefficients are set as a function of the impact angle, θ_1, in *Figure 25.19* (p. 1051).

Note that the default setting for both coefficients of restitution is a constant value of 1.0 (all normal and tangential momentum retained).

- **trap**

The trajectory calculations are terminated and the fate of the particle is recorded as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051). In the case of combusting particles, the remaining volatile mass is passed into the vapor phase.

Figure 25.20 "Trap" Boundary Condition for the Discrete Phase

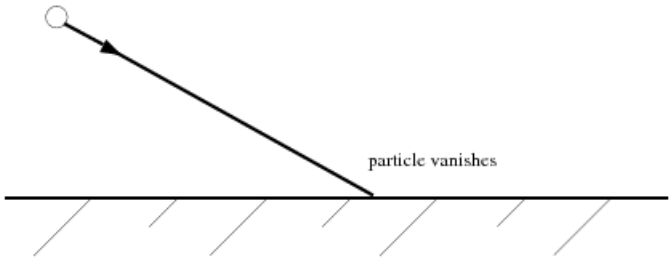


- **escape**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The particle is reported as having "escaped" when it encounters the boundary in question. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

Figure 25.21 "Escape" Boundary Condition for the Discrete Phase



- **wall-jet**

  The **wall-jet** type boundary condition is appropriate for high-temperature walls where no significant liquid film is formed, and in high-Weber-number impacts where the spray acts as a jet. The model is not appropriate for regimes where film is important (e.g., port fuel injection in SI engines, rainwater runoff, etc.).

  A more detailed description of underlying theory is available in Wall-Jet Model Theory in the Theory Guide.

- **wall-film**

  This boundary condition consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide.

  Important

  Note that the **Workpile Algorithm** option is not available with the wall film boundary condition. It will be disabled automatically when choosing to simulate a wall film on a wall.

- **interior**

  This boundary condition means that the particles will pass through the internal boundary. This option is available only for internal boundary zones, such as a radiator or a porous jump.

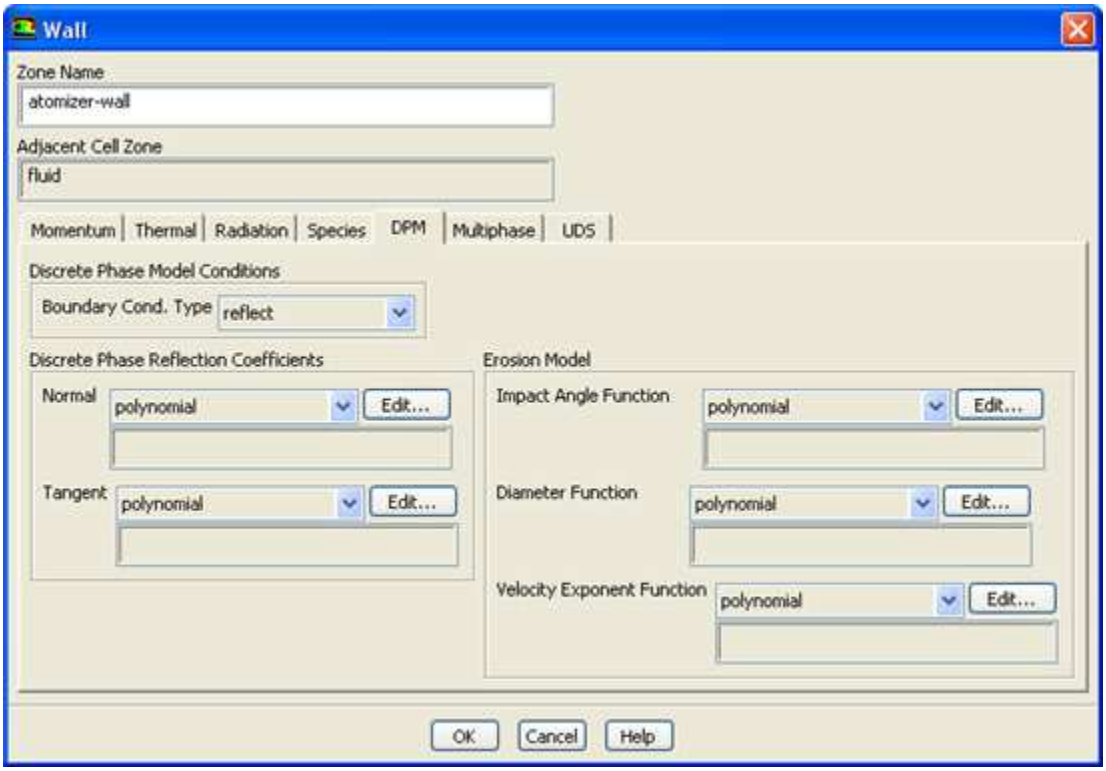
It is also possible to use a user-defined function to compute the behavior of the particles at a physical boundary. More information about user-defined functions can be found in the UDF Manual.

Because you can stipulate any of these conditions at flow boundaries, it is possible to incorporate mixed discrete phase boundary conditions in your ANSYS FLUENT model.

Discrete phase boundary conditions can be set for boundaries in the dialog boxes opened from the **Boundary Conditions** task page. When one or more injections have been defined, inputs for the discrete phase will appear in the dialog boxes (e.g., *Figure 25.22* (p. 1053)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.22 Discrete Phase Boundary Conditions in the Wall Dialog Box



Select **reflect**, **trap**, **escape**, **wall-jet**, **wall-film**, **interior**, or **user-defined** from the **Boundary Cond. Type** drop-down list under **Discrete Phase Model Conditions**, as shown in *Figure 25.22* (p. 1053). (In the **Walls** dialog boxes, you will need to click the **DPM** tab to access the **Discrete Phase Model Conditions**.) If you select **user-defined**, you can select a user-defined function in the **Boundary Cond. Function** drop-down list. For internal boundary zones, such as a radiator or a porous jump, you can also choose an **interior** boundary condition. The **interior** condition means that the particles will pass through the internal boundary.

If you select the **reflect** type at a wall (only), you can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Normal** and **Tangent** coefficients of restitution under **Discrete Phase Reflection Coefficients**. See *Discrete Phase Boundary Condition Types* (p. 1050) for details about the boundary condition types and the coefficients of restitution. The dialog boxes for defining the polynomial, piecewise-linear, and piecewise-polynomial functions are the same as those used for defining temperature-dependent properties. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details.

25.4.1.1. Default Discrete Phase Boundary Conditions

ANSYS FLUENT makes the following assumptions regarding boundary conditions:

- The **reflect** type is assumed at wall, symmetry, and axis boundaries, with both coefficients of restitution equal to 1.0
- The **escape** type is assumed at all flow boundaries (pressure and velocity inlets, pressure outlets, etc.)
- The **interior** type is assumed at all internal boundaries (radiator, porous jump, etc.)

The coefficient of restitution can be modified only for wall boundaries.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.4.2. Setting Particle Erosion and Accretion Parameters

If the **Erosion/Accretion** option is selected in the *Discrete Phase Model Dialog Box* (p. 1723), the erosion rate expression must be specified at the walls. The erosion rate is defined in Equation 16–205 in the Theory Guide as a product of the mass flux and specified functions for the particle diameter, impact angle, and velocity exponent. Under **Erosion Model** in the *Wall Dialog Box* (p. 1865), you can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Impact Angle Function**, **Diameter Function**, and **Velocity Exponent Function** ($f(\alpha)$, $C\left(d_p\right)$, and $b(v)$ in Equation 16–205 in the Theory Guide). See Particle Erosion and Accretion Theory in the Theory Guide and *Monitoring Erosion/Accretion of Particles at Walls* (p. 1017) for a detailed description of these functions and *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details about using the dialog boxes for defining polynomial, piecewise-linear, and piecewise-polynomial functions.

25.5. Setting Material Properties for the Discrete Phase

In order to apply the physical models described in earlier sections to the prediction of the discrete phase trajectories and heat/mass transfer, ANSYS FLUENT requires many physical property inputs.

For additional information, please see the following sections:

25.5.1. Summary of Property Inputs
25.5.2. Setting Discrete-Phase Physical Properties

25.5.1. Summary of Property Inputs

Table 25.1: Property Inputs for Inert Particles (p. 1054) – *Table 25.5: Property Inputs for Multicomponent Particles (Law 7)* (p. 1057) summarize which of these property inputs are used for each particle type and in which of the equations for heat and mass transfer each property input is used. Detailed descriptions of each input are provided in *Setting Discrete-Phase Physical Properties* (p. 1057).

Table 25.1 Property Inputs for Inert Particles

Property	Symbol
density	ρ_p in Equation 16–1 in the Theory Guide
specific heat	c_p in Equation 16–74
particle emissivity	ε_p in Equation 16–74
particle scattering factor	f in Equation 5–34
thermophoretic coefficient	$D_{T,p}$ in Equation 16–8

Table 25.2 Property Inputs for Droplet Particles

Properties	Symbol
density	ρ_p in Equation 16–1 in the Theory Guide
specific heat	c_p in Equation 16–88
thermal conductivity	k_p in Equation 16–9
viscosity	μ in Equation 16–252

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties	Symbol
latent heat	h_{fg} in Equation 16–88
vaporization temperature	T_{vap} in Equation 16–81
boiling point	T_{bp} in Equation 16–81, Equation 16–89
volatile component fraction	f_{v0} in Equation 16–82, Equation 16–90
binary diffusivity	$D_{i,m}$ in Equation 16–86
saturation vapor pressure	$p_{sat}(T)$ in Equation 16–84
heat of pyrolysis	h_{pyrol} in Equation 16–301
droplet surface tension	σ in Equation 16–226, Equation 16–251
particle emissivity	ε_p in Equation 16–88, Equation 16–94
particle scattering factor	f in Equation 5–34
thermophoretic coefficient	$D_{T,p}$ in Equation 16–8
critical temperature	$T_{c,i}$ in Equation 16–165
critical pressure	$p_{c,i}$ in Equation 16–165
accentric factor	ω_i in Equation 16–165

Table 25.3 Property Inputs for Combusting Particles (Laws 1–4)

Properties	Symbol
density	ρ_p in Equation 16–1 in the Theory Guide
specific heat	c_p in Equation 16–74
latent heat	h_{fg} in Equation 16–301
vaporization temperature	$T_{vap}=T_{bp}$ in Equation 16–95
volatile component fraction	f_{v0} in Equation 16–96
swelling coefficient	C_{sw} in Equation 16–128
burnout stoichiometric ratio	S_b in Equation 16–135
combustible fraction	f_{comb} in Equation 16–134
heat of reaction for burnout	H_{reac} in Equation 16–135 Equation 16–149
fraction of reaction heat given to solid	f_h in Equation 16–149
particle emissivity	ε_p in Equation 16–129, Equation 16–149
particle scattering factor	f in Equation 5–34
thermophoretic coefficient	$D_{T,p}$ in Equation 16–8
devolatilization model	
– law 4, constant rate	

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties	Symbol
– – constant	A_0 in Equation 16–97
– law 4, single rate	
– – pre-exponential factor	A_1 in Equation 16–98
– – activation energy	E in Equation 16–98
– law 4, two rates	
– – pre-exponential factors	A_1, A_2 in Equation 16–101, Equation 16–102
– – activation energies	E_1, E_2 in Equation 16–101, Equation 16–102
– – weighting factors	α_1, α_2 in Equation 16–103
– law 4, CPD	
– – initial fraction of bridges in coal lattice	p_0 in Equation 16–114
– – initial fraction of char bridges	c_0 in Equation 16–113
– – lattice coordination number	$\sigma + 1$ in Equation 16–125
– – cluster molecular weight	$M_{w,1}$ in Equation 16–125
– – side chain molecular weight	$M_{w,\delta}$ in

Table 25.4 Property Inputs for Combusting Particles (Law 5)

Properties	Symbol
combustion model	
– law 5, diffusion rate	
– – binary diffusivity	$D_{i,m}$ in Equation 16–136 in the Theory Guide
– law 5, diffusion/kinetic rate	
– – mass diffusion limited rate constant	C_1 in Equation 16–137
– – kinetics limited rate pre-exp. factor	C_2 in Equation 16–138
– – kinetics limited rate activation. energy	E in Equation 16–138
– law 5, intrinsic rate	
– – mass diffusion limited rate constant	C_1 in Equation 16–137
– – kinetics limited rate pre-exp. factor	A_i in Equation 16–147
– – kinetics limited rate activ. energy	E_i in Equation 16–147
– – char porosity	θ in Equation 16–144
– – mean pore radius	$\bar{r}_p$ in Equation 16–146
– – specific internal surface area	A_g in Equation 16–141, Equation 16–143
– – tortuosity	τ in Equation 16–144
– – burning mode	α in Equation 16–148
– law 5, multiple surface reaction	

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties	Symbol
– – binary diffusivity	$D_{i,m}$ in Equation 16–136

Table 25.5 Property Inputs for Multicomponent Particles (Law 7)

Property	Symbol
mixture species	selected droplets for components
density	ρ_p in Equation 16–1 of the Theory Guide
specific heat	c_p in Equation 16–152
thermal conductivity	k_p in Equation 16–9
vapor particle equilibrium	$C_{i,s}$ in Equation 16–160

25.5.2. Setting Discrete-Phase Physical Properties

25.5.2.1. The Concept of Discrete-Phase Materials

When you create a particle injection and define the initial conditions for the discrete phase (as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022)), you choose a particular material as the particle's material. All particle streams of that material will have the same physical properties.

Important

Note that you will not choose a **Material** for a **Massless** particle type in the **Set Injections Properties** dialog box.

Discrete-phase materials are divided into four categories, corresponding to the four types of particles available. These material types are **inert-particle**, **droplet-particle**, **combusting-particle**, and **multicomponent-particle**. Each material type will be added to the **Material Type** list in the *Create/Edit Materials Dialog Box* (p. 1736) when an injection of that type of particle is defined (in the **Set Injection Properties** or **Set Multiple Injection Properties** dialog box, as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022)). The first time you create an injection of each particle type, you will be able to choose a material from the database, and this will become the default material for that type of particle. That is, if you create another injection of the same type of particle, your selected material will be used for that injection as well. You may choose to modify the predefined properties for your selected particle material, if you want (as described in *Modifying Properties of an Existing Material* (p. 388)). If you need only one set of properties for each type of particle, you need not define any new materials; you can simply use the same material for all particles.

Important

If you do not find the material you want in the database, you can select a material that is close to the one you wish to use, and then modify the properties and give the material a new name, as described in *Creating a New Material* (p. 390).

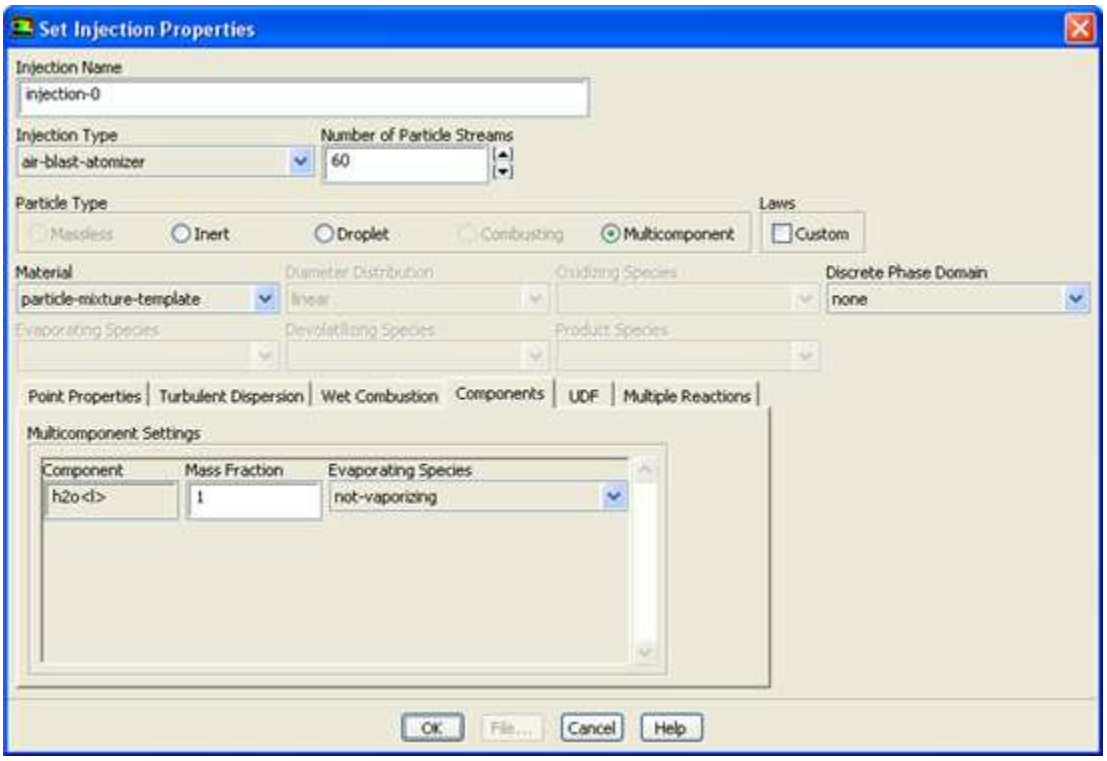
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that a discrete-phase material type will not appear in the **Material Type** list in the **Create/Edit Materials** dialog boxes until you have defined an injection of that type of particles. This means, for example, that you cannot define or modify any combusting-particle materials until you have defined a combusting particle injection (as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022)).

For a **particle-mixture** material type, you will need to select the species in your mixture. To do this, click the **Edit...** button next to **Mixture Species** in the **Create/Edit Materials** dialog box. The **Species** dialog box will open, where you will include your **Selected Species**. The selected species will now be available in the **Set Injection Properties** dialog box, under the **Components** tab (*Figure 25.23* (p. 1058)).

Figure 25.23 The Components Tab



25.5.2.1.1. Defining Additional Discrete-Phase Materials

In many cases, a single set of physical properties (density, heat capacity, etc.) is appropriate for each type of discrete phase particle considered in a given model. Sometimes, however, a single model may contain two different types of inert, droplet, combusting particles, or multicomponent particles (e.g., heavy particles and gaseous bubbles or two different types of evaporating liquid droplets). In such cases, it is necessary to assign a different set of properties to the two (or more) different types of particles. This is easily accomplished by defining two or more inert, droplet, or combusting particle materials and using the appropriate one for each particle injection.

You can define additional discrete-phase materials either by copying them from the database or by creating them from scratch. See *Using the Materials Task Page* (p. 387) for instructions on using the *Create/Edit Materials Dialog Box* (p. 1736) to perform these actions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Recall that you must define at least one injection (as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022)) containing particles of a certain type before you will be able to define additional materials for that particle type.

25.5.2.2. Description of the Properties

The properties that appear in the *Create/Edit Materials Dialog Box* (p. 1736) vary depending on the particle type (selected in the **Set Injection Properties** or **Set Multiple Injection Properties** dialog box, as described in *Defining Injection Properties* (p. 1040) and *Defining Properties Common to More than One Injection* (p. 1047)) and the physical models you are using in conjunction with the discrete-phase model.

Below, all properties you may need to define for a discrete-phase material are listed. See *Table 25.1: Property Inputs for Inert Particles* (p. 1054) – *Table 25.4: Property Inputs for Combusting Particles (Law 5)* (p. 1056) to see which properties are defined for each type of particle.

Density

is the density of the particulate phase in units of mass per unit volume of the discrete phase. This density is the mass density and not the volumetric density. Since certain particles may swell during the trajectory calculations, your input is actually an "initial" density.

Cp

is the specific heat, c_p, of the particle. The specific heat may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Cp**. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details about temperature-dependent properties. For multicomponent particles, it can be calculated as a mass-weighted value of the specific heat of the droplet component.

Thermal Conductivity

is the thermal conductivity of the particle. This input is specified in units of W/m-K in SI units or Btu/ft-h-$^\circ$F in British units and is treated as a constant by ANSYS FLUENT.

Latent Heat

is the latent heat of vaporization, h_{fg}, required for phase change from an evaporating liquid droplet (Equation 16–88 in the Theory Guide) or for the evolution of volatiles from a combusting particle (Equation 16–129 in the Theory Guide). This input is supplied in units of J/kg in SI units or of Btu/lb_m in British units and is treated as a constant by ANSYS FLUENT. For the droplet particle, the latent heat value at the boiling point temperature should be used.

Thermophoretic Coefficient

is the coefficient $D_{T,p}$ in Equation 16–8 in the Theory Guide, and appears when the thermophoretic force (which is described in Thermophoretic Force in the Theory Guide) is included in the trajectory calculation (i.e., when the **Thermophoretic Force** option is enabled in the **Discrete Phase Model** dialog box). The default is the expression developed by Talbot [*90*] (p. 2213) (**talbot-diffusion-coeff**) and requires no input from you. You can also define the thermophoretic coefficient as a function of temperature by selecting one of the function types from the drop-down list to the right of **Thermophoretic Coefficient**. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details about temperature-dependent properties.

Vaporization Temperature

is the temperature, T_{vap}, at which the calculation of vaporization from a liquid droplet or devolatilization from a combusting particle is initiated by ANSYS FLUENT. Until the particle temperature reaches T_{vap},

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the particle is heated via Law 1, Equation 16–74 in the Theory Guide. This temperature input represents a modeling decision rather than any physical characteristic of the discrete phase.

Boiling Point

is the temperature, T_{bp}, at which the calculation of the boiling rate equation (Equation 16–91 in the Theory Guide) is initiated by ANSYS FLUENT. When a droplet particle reaches the boiling point, ANSYS FLUENT applies Law 3 and assumes that the droplet temperature is constant at T_{bp}. The boiling point denotes the temperature at which the particle law transitions from the vaporization law to the boiling law.

For multicomponent particles the boiling point of the components is used only as a reference temperature of the latent heat. Instead, the boiling starts when the sum of the partial component saturation pressures reach the total fluid pressure. The definition of the saturation pressure curve is therefore essential for the boiling of multicomponent particles.

Vapor-Particle-Equilibrium

is the selected approach for the calculation of the vapor concentration of the components at the surface. This can be Raoult's law (Equation 16–159 in the Theory Guide), the Peng-Robinson real gas model (Equation 16–168 in the Theory Guide), or a user-defined function that provides this value.

Critical Temperature

is the temperature $T_{c,i}$ (Equation 16–165 in the Theory Guide) of the pure component for multicomponent particles when using the Peng-Robinson real gas model for calculating the vapor-particle-equilibrium.

Critical Pressure

is the pressure $p_{c,i}$ (Equation 16–165 in the Theory Guide) of the pure component for multicomponent particles when using the Peng-Robinson real gas model for calculating the vapor-particle-equilibrium.

Accentric Factor

is the constant ω_i (Equation 16–165 in the Theory Guide) of the pure component for multicomponent particles when using the Peng-Robinson real gas model for calculating the vapor-particle-equilibrium.

Volatile Component Fraction

(f_{v0}) is the fraction of a droplet particle that may vaporize via Laws 2 and/or 3 (Droplet Vaporization (Law 2) in the Theory Guide). For combusting particles, it is the fraction of volatiles that may be evolved via Law 4 (Devolatilization (Law 4) in the Theory Guide).

Binary Diffusivity

is the mass diffusion coefficient, $D_{i,m}$, used in the vaporization law, Law 2 (Equation 16–86 in the Theory Guide). This input is also used to define the mass diffusion of the oxidizing species to the surface of a combusting particle, $D_{i,m}$, as given in Equation 16–136 in the Theory Guide. (Note that the diffusion coefficient inputs that you supply for the continuous phase are not used for the discrete phase.)

Saturation Vapor Pressure

is the saturated vapor pressure, p_{sat}, defined as a function of temperature, which is used in the vaporization law, Law 2 (Equation 16–84 in the Theory Guide). The saturated vapor pressure may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of its name. (See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details about temperature-dependent properties.) In the case of unrealistic inputs, ANSYS FLUENT restricts the range of P_{sat} to between 0.0 and the operating pressure. Correct input of a realistic vapor pressure curve is essential for accurate results from the vaporization model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat of Pyrolysis

is the heat of the instantaneous pyrolysis reaction , h_{pyrol}, that the evaporating/boiling species may undergo when released to the continuous phase. This input represents the conversion of the evaporating species to lighter components during the evaporation process. The heat of pyrolysis should be input as a positive number for exothermic reaction and as a negative number for endothermic reaction. The default value of zero implies that the heat of pyrolysis is not considered. This input is used in Equation 16–301 in the Theory Guide.

Swelling Coefficient

is the coefficient C_{sw} in Equation 16–128 in the Theory Guide, which governs the swelling of the coal particle during the devolatilization law, Law 4 (Devolatilization (Law 4) in the Theory Guide). A swelling coefficient of unity (the default) implies that the coal particle stays at constant diameter during the devolatilization process.

Burnout Stoichiometric Ratio

is the stoichiometric requirement, S_b, for the burnout reaction, Equation 16–135 in the Theory Guide, in terms of mass of oxidant per mass of char in the particle.

Combustible Fraction

is the mass fraction of char, f_{comb}, in the coal particle, i.e., the fraction of the initial combusting particle that will react in the surface reaction, Law 5 (Equation 16–134 in the Theory Guide).

Heat of Reaction for Burnout

is the heat released by the surface char combustion reaction, Law 5 (Equation 16–135 in the Theory Guide). This parameter is input in terms of heat release (e.g., Joules) per unit mass of char consumed in the surface reaction.

React. Heat Fraction Absorbed by Solid

is the parameter f_h (Equation 16–149 in the Theory Guide), which controls the distribution of the heat of reaction between the particle and the continuous phase. The default value of zero implies that the entire heat of reaction is released to the continuous phase.

Devolatilization Model

defines which version of the devolatilization model, Law 4, is being used. If you want to use the default constant rate devolatilization model, Equation 16–97 in the Theory Guide, retain the selection of **constant** in the drop-down list to the right of **Devolatilization Model** and input the rate constant A_0 in the field below the list.

You can activate one of the optional devolatilization models (the single kinetic rate, two kinetic rates, or CPD model, as described in Devolatilization (Law 4) in the Theory Guide) by choosing **single rate**, **two-competing-rates**, or **cpd-model** in the drop-down list.

When the single kinetic rate model (**single-rate**) is selected, the *Single Rate Devolatilization Dialog Box* (p. 1778) will appear and you will enter the **Pre-exponential Factor**, A_1, and the **Activation Energy**, E, to be used in Equation 16–99 in the Theory Guide for the computation of the kinetic rate.

When the two competing rates model (**two-competing-rates**) is selected, the *Two Competing Rates Model Dialog Box* (p. 1779) will appear and you will enter, for the **First Rate** and the **Second Rate**, the **Pre-exponential Factor** (A_1 in Equation 16–101 and A_2 in Equation 16–102 in the Theory Guide), **Activation Energy** (E_1 in Equation 16–101 and E_2 in Equation 16–102), and **Weighting Factor** (α_1 and α_2 in Equation 16–103). The constants you input are used in Equation 16–101 through Equation 16–103.

When the CPD model (**cpd-model**) is selected, the **CPD Model** dialog box will appear and you will enter the **Initial Fraction of Bridges in Coal Lattice** (p_0 in Equation 16–114 of the Theory Guide), **Initial**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fraction of Char Bridges (c_0 in Equation 16–113), **Lattice Coordination Number** ($\sigma + 1$ in Equation 16–125), **Cluster Molecular Weight** ($M_{w,1}$ in Equation 16–125), and **Side Chain Molecular Weight** ($M_{w,\delta}$ in Equation 16–124).

Note that the **Single Rate Devolatilization Model**, **Two Competing Rates Model**, and **CPD Model** dialog boxes are modal dialog boxes, which means that you must tend to them immediately before continuing the property definitions.

Combustion Model

defines which version of the surface char combustion law (Law5) is being used. If you want touse the default diffusion-limited rate model, retain the selection of **diffusion-limited** in the drop-down list to the right of **Combustion Model**. No additional inputs are necessary, because the binary diffusivity defined above will be used in Equation 16–136 in the Theory Guide.

To use the kinetics/diffusion-limited rate model for the surface combustion model, select **kinetics/diffusion-limited** in the drop-down list. The *Kinetics/Diffusion-Limited Combustion Model Dialog Box* (p. 1780) will appear and you will enter the **Mass Diffusion Limited Rate Constant** (C_1 in Equation 16–137 in the Theory Guide), **Kinetics Limited Rate Pre-exponential Factor** (C_2 in Equation 16–138), and **Kinetics Limited Rate Activation Energy** (E in Equation 16–138).

Note that the **Kinetics/Diffusion-Limited Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

To use the intrinsic model for the surface combustion model, select **intrinsic-model** in the drop-down list. The *Intrinsic Combustion Model Dialog Box* (p. 1781) will appear and you will enter the **Mass Diffusion Limited Rate Constant** (C_1 in Equation 16–137 in the Theory Guide), **Kinetics Limited Rate Pre-exponential Factor** (A_i in Equation 16–147), **Kinetics Limited Rate Activation Energy** (E_i in Equation 16–147), **Char Porosity** (θ in Equation 16–144), **Mean Pore Radius** ($\bar{r}_p$ in Equation 16–146), **Specific Internal Surface Area** (A_g in Equation 16–141 Equation 16–143), **Tortuosity** (τ in Equation 16–144), and **Burning Mode, alpha** (α in Equation 16–148).

Note that the **Intrinsic Combustion Model** dialog box is a model dialog box, which means that you must tend to it immediately before continuing the property definitions.

To use the multiple surface reactions model, select **multiple-surface-reactions** in the drop-down list. ANSYS FLUENT will display a dialog box informing you that you will need to open the **Reactions** dialog box, where you can review or modify the particle surface reactions that you specified as described in *Overview of User Inputs for Modeling Species Transport and Reactions* (p. 799).

Important

If you have not yet defined any particle surface reactions, you must be sure to define them now. See *Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion* (p. 836) for more information about using the multiple surface reactions model.

You will notice that the **Burnout Stoichiometric Ratio** and **Heat of Reaction for Burnout** are no longer available in the **Create/Edit Materials** dialog box, as these parameters are now computed from the particle surface reactions you defined in the **Reactions** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the multiple surface reactions model is available only if the **Particle Surface** option for **Reactions** is enabled in the **Species Model** dialog box. See *User Inputs for Particle Surface Reactions* (p. 835) for details.

When the effect of particles on radiation is enabled (for the P-1 or discrete ordinates radiation model only) in the *Discrete Phase Model Dialog Box* (p. 1723), you will need to define the following additional parameters:

Particle Emissivity

is the emissivity of particles in your model, ε_p, used to compute radiation heat transfer to the particles (Equation 16–74, Equation 16–88, Equation 16–94, Equation 16–129, and Equation 16–149 in the Theory Guide) when the P-1 or discrete ordinates radiation model is active. Note that you must enable radiation to particles, using the **Particle Radiation Interaction** option in the *Discrete Phase Model Dialog Box* (p. 1723). Recommended values of particle emissivity are 1.0 for coal particles and 0.5 for ash [*47*] (p. 2211).

Particle Scattering Factor

is the scattering factor, f_p, due to particles in the P-1 or discrete ordinates radiation model (Equation 5–34 in the Theory Guide). Note that you must enable particle effects in the radiation model, using the **Particle Radiation Interaction** option in the *Discrete Phase Model Dialog Box* (p. 1723). The recommended value of f_p for coal combustion modeling is 0.9 [*47*] (p. 2211). Note that if the effect of particles on radiation is enabled, scattering in the continuous phase will be ignored in the radiation model.

When an atomizer injection model and/or the droplet breakup or collision model is enabled in the *Set Injection Properties Dialog Box* (p. 2097) (atomizers) and/or *Discrete Phase Model Dialog Box* (p. 1723) (droplet breakup/collision), you will need to define the following additional parameters:

Viscosity

is the droplet viscosity, μ_l. The viscosity may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Viscosity**. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details about temperature-dependent properties. You also have the option of implementing a user-defined function to model the droplet viscosity. More information about user-defined functions can be found in the UDF Manual.

Droplet Surface Tension

is the droplet surface tension, σ. The surface tension may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Droplet Surface Tension**. See *Defining Properties Using Temperature-Dependent Functions* (p. 398) for details about temperature dependent properties. You also have the option of implementing a user-defined function to model the droplet surface tension. More information about user-defined functions can be found in the UDF Manual.

25.6. Solution Strategies for the Discrete Phase

Solution of the discrete phase implies integration in time of the force balance on the particle (Equation 16–1 in the Theory Guide) to yield the particle trajectory. As the particle is moved along its trajectory, heat and mass transfer between the particle and the continuous phase are also computed via the heat/mass transfer laws (Laws for Heat and Mass Exchange in the Theory Guide). The accuracy of the discrete phase calculation thus depends on the time accuracy of the integration and upon the appropriate coupling between the discrete and continuous phases when required. Numerical controls are described in *Numerics of the Discrete Phase Model* (p. 1019). Coupling and performing trajectory calculations are described in *Performing Trajectory Calculations* (p. 1064). *Resetting the Interphase Exchange Terms* (p. 1067) and *Parallel Processing for the Discrete Phase Model* (p. 1087) provide information about resetting interphase exchange terms and using the parallel solver for a discrete phase calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, please see the following sections:

25.6.1. Performing Trajectory Calculations
25.6.2. Resetting the Interphase Exchange Terms

25.6.1. Performing Trajectory Calculations

The trajectories of your discrete phase injections are computed when you display the trajectories using graphics or when you perform solution iterations. That is, you can display trajectories without impacting the continuous phase, or you can include their effect on the continuum (termed a coupled calculation). In turbulent flows, trajectories can be based on mean (time-averaged) continuous phase velocities or they can be impacted by instantaneous velocity fluctuations in the fluid. This section describes the procedures and commands you use to perform coupled or uncoupled trajectory calculations, with or without stochastic tracking or cloud tracking.

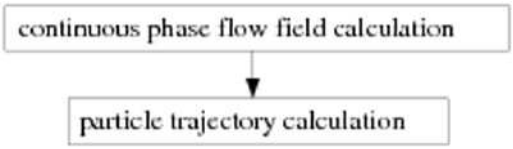
25.6.1.1. Uncoupled Calculations

For the uncoupled calculation, you will perform the following two steps:

1. Solve the continuous phase flow field.
2. Plot (and report) the particle trajectories for discrete phase injections of interest.

In the uncoupled approach, this two-step procedure completes the modeling effort, as illustrated in *Figure 25.24* (p. 1064). The particle trajectories are computed as they are displayed, based on a fixed continuous-phase flow field. Graphical and reporting options are detailed in *Postprocessing for the Discrete Phase* (p. 1067).

Figure 25.24 Uncoupled Discrete Phase Calculations



This procedure is adequate when the discrete phase is present at a low mass and momentum loading, in which case the continuous phase is not impacted by the presence of the discrete phase.

25.6.1.2. Coupled Calculations

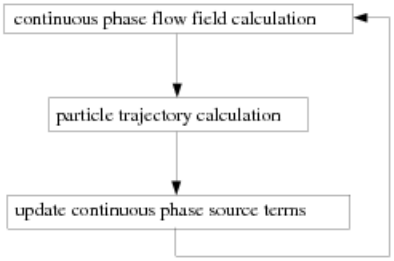
In a coupled two-phase simulation, ANSYS FLUENT modifies the two-step procedure above as follows:

1. Solve the continuous phase flow field (prior to introduction of the discrete phase).
2. Introduce the discrete phase by calculating the particle trajectories for each discrete phase injection.
3. Recalculate the continuous phase flow, using the interphase exchange of momentum, heat, and mass determined during the previous particle calculation.
4. Recalculate the discrete phase trajectories in the modified continuous phase flow field.
5. Repeat the previous two steps until a converged solution is achieved in which both the continuous phase flow field and the discrete phase particle trajectories are unchanged with each additional calculation.

This coupled calculation procedure is illustrated in *Figure 25.25* (p. 1065). When your ANSYS FLUENT model includes a high mass and/or momentum loading in the discrete phase, the coupled procedure must be followed in order to include the important impact of the discrete phase on the continuous phase flow field.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.25 Coupled Discrete Phase Calculations



Important

When you perform coupled calculations, all defined discrete phase injections will be computed. You cannot calculate a subset of the injections you have defined. If there are massless particle injections defined, these will have no effect in the coupled calculation.

25.6.1.2.1. Procedures for a Coupled Two-Phase Flow

If your ANSYS FLUENT model includes prediction of a coupled two-phase flow, you should begin with a partially (or fully) converged continuous-phase flow field. You will then create your injection(s) and set up the coupled calculation.

For each discrete-phase iteration, ANSYS FLUENT computes the particle/droplet trajectories and updates the interphase exchange of momentum, heat, and mass in each control volume. These interphase exchange terms then impact the continuous phase when the continuous phase iteration is performed. During the coupled calculation, ANSYS FLUENT will perform the discrete phase iteration at specified intervals during the continuous-phase calculation. The coupled calculation continues until the continuous phase flow field no longer changes with further calculations (i.e., all convergence criteria are satisfied). When convergence is reached, the discrete phase trajectories no longer change either, since changes in the discrete phase trajectories would result in changes in the continuous phase flow field.

The steps for setting up the coupled calculation are as follows:

1. Solve the continuous phase flow field.
2. In the *Discrete Phase Model Dialog Box* (p. 1723) (*Figure 25.1* (p. 1013)), enable the **Interaction with Continuous Phase** option.
3. Set the frequency with which the particle trajectory calculations are introduced in the **Number of Continuous Phase Iterations Per DPM Iteration** field. If you set this parameter to 5, for example, a discrete phase iteration will be performed every fifth continuous phase iteration. The optimum number of iterations between trajectory calculations depends upon the physics of your ANSYS FLUENT model.

   Important

   Note that if you set this parameter to 0, ANSYS FLUENT will not perform any discrete phase iterations.

During the coupled calculation (which you initiate using the *Run Calculation Task Page* (p. 1955) in the usual manner) you will see the following information in the ANSYS FLUENT console as the continuous and discrete phase iterations are performed:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 iter   continuity   x-velocity   y-velocity       k         epsilon       energy    time/it
  314   2.5249e-01   2.8657e-01   1.0533e+00   7.6227e-02   2.9771e-02   9.8181e-03   :00:05
  315   2.7955e-01   2.5867e-01   9.2736e-01   6.4516e-02   2.6545e-02   4.2314e-03   :00:03

   DPM  Iteration  ....
   number  tracked=  9,  number  escaped=  1,  aborted=  0,  trapped=  0,  evaporated  =  8,i
 Done.
  316   1.9206e-01   1.1860e-01   6.9573e-01   5.2692e-02   2.3997e-02   2.4532e-03   :00:02
  317   2.0729e-01   3.2982e-02   8.3036e-01   4.1649e-02   2.2111e-02   2.5369e-01   :00:01
  318   3.2820e-01   5.5508e-02   6.0900e-01   5.9018e-02   2.6619e-02   4.0394e-02   :00:00
```

Note that you can perform a discrete phase calculation at any time by using the `solve/dpm-update` text command.

25.6.1.2.2. Stochastic Tracking in Coupled Calculations

If you include the stochastic prediction of turbulent dispersion in the coupled two-phase flow calculations, the number of stochastic tries applied each time the discrete phase trajectories are introduced during coupled calculations will be equal to the **Number of Tries** specified in the *Set Injection Properties Dialog Box* (p. 2097). Input of this parameter is described in *Stochastic Tracking* (p. 1044).

Note that the number of tries should be set to 0 if you want to perform the coupled calculation based on the mean continuous phase flow field. An input of $n \geq 1$ requests n stochastic trajectory calculations for each particle in the injection. Note that when the number of stochastic tracks included is small, you may find that the ensemble average of the trajectories is quite different each time the trajectories are computed. These differences may, in turn, impact the convergence of your coupled solution. For this reason, you should include an adequate number of stochastic tracks in order to avoid convergence troubles in coupled calculations.

25.6.1.2.3. Under-Relaxation of the Interphase Exchange Terms

When you are coupling the discrete and continuous phases for steady-state calculations, using the calculation procedures noted above, ANSYS FLUENT applies under-relaxation to the momentum, heat, and mass transfer terms. This under-relaxation serves to increase the stability of the coupled calculation procedure by letting the impact of the discrete phase change only gradually:

$$E_{new} = E_{old} + \alpha \left(E_{calculated} - E_{old} \right) \qquad \textbf{(25–10)}$$

where E_{new} is the exchange term, E_{old} is the previous value, $E_{calculated}$ is the newly computed value, and α is the particle/droplet under-relaxation factor. ANSYS FLUENT uses a default value of 0.5 for α. You can modify $\alpha$by changing the value in the **Discrete Phase Sources** field under **Under-Relaxation Factors** in the **Solution Controls** task page. You may need to decrease α in order to improve the stability of coupled discrete phase calculations.

Figure 25.26 (p. 1067) shows how the source term, S, when applied to the flow equations, changes with the number of updates for varying under-relaxation factors. In *Figure 25.26* (p. 1067), S_∞ is the final source term for which a value is reached after a certain number of updates and S_0 is the initial source term at the start of the computation. The value of S_0 is typically zero at the beginning of the calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.26 Effect of Number of Source Term Updates on Source Term Applied to Flow Equations

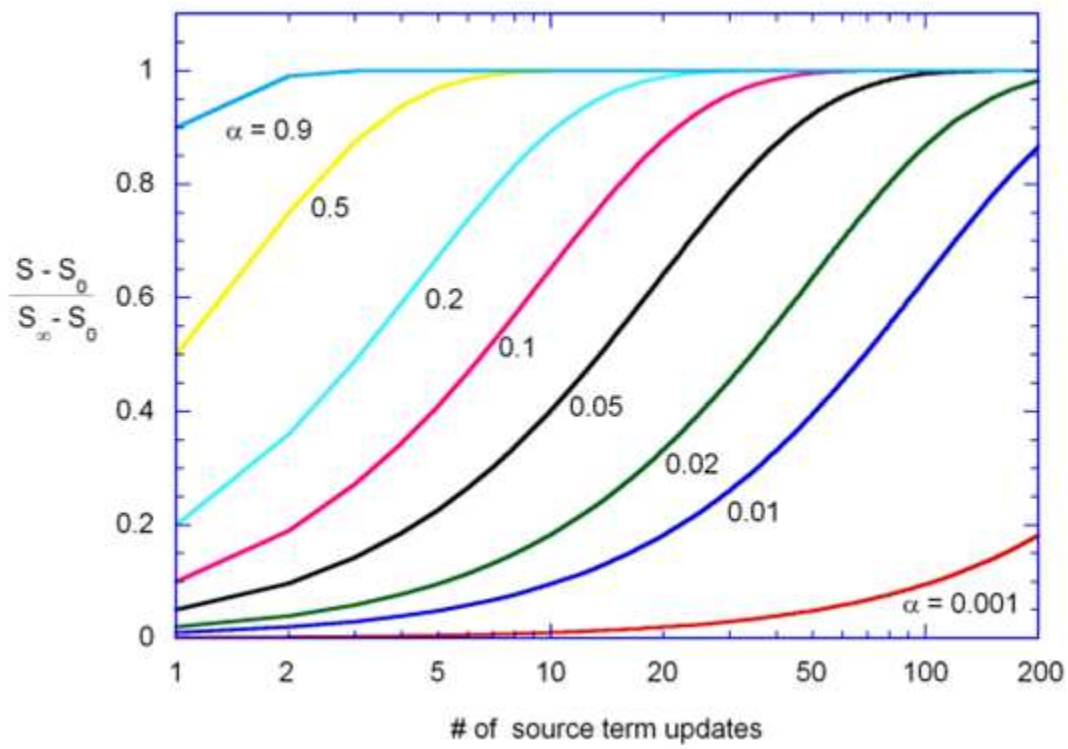


Figure 25.26 (p. 1067) can be applied to this option as well. Keep in mind that the DPM source terms are updated every continuous flow iteration.

25.6.2. Resetting the Interphase Exchange Terms

If you have performed coupled calculations, resulting in nonzero interphase sources/sinks of momentum, heat, and/or mass that you do not want to include in subsequent calculations, you can reset these sources to zero.

Solution Initialization → **Reset DPM Sources**

When you click the **Reset DPM Sources** button, the sources will immediately be reset to zero without any further confirmation from you.

25.7. Postprocessing for the Discrete Phase

After you have completed your discrete phase inputs and any coupled two-phase calculations of interest, you can display and store the particle trajectory predictions. ANSYS FLUENT provides both graphical and alphanumeric reporting facilities for the discrete phase, including the following:

- graphical display of the particle trajectories
- summary reports of trajectory fates
- step-by-step reports of the particle position, velocity, temperature, and diameter
- alphanumeric reports and graphical display of the interphase exchange of momentum, heat, and mass
- sampling of trajectories at boundaries and lines/planes

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- summary reporting of current particles in the domain
- histograms of trajectory data at sample planes
- display of erosion/accretion rates
- exporting of trajectories to Fieldview and Ensight

This section provides detailed descriptions of each of these postprocessing options.

(Note that plotting or reporting trajectories does not change the source terms.)

For additional information, please see the following sections:

25.7.1. Displaying of Trajectories
25.7.2. Reporting of Trajectory Fates
25.7.3. Step-by-Step Reporting of Trajectories
25.7.4. Reporting of Current Positions for Unsteady Tracking
25.7.5. Reporting of Interphase Exchange Terms and Discrete Phase Concentration
25.7.6. Sampling of Trajectories
25.7.7. Histogram Reporting of Samples
25.7.8. Summary Reporting of Current Particles
25.7.9. Postprocessing of Erosion/Accretion Rates

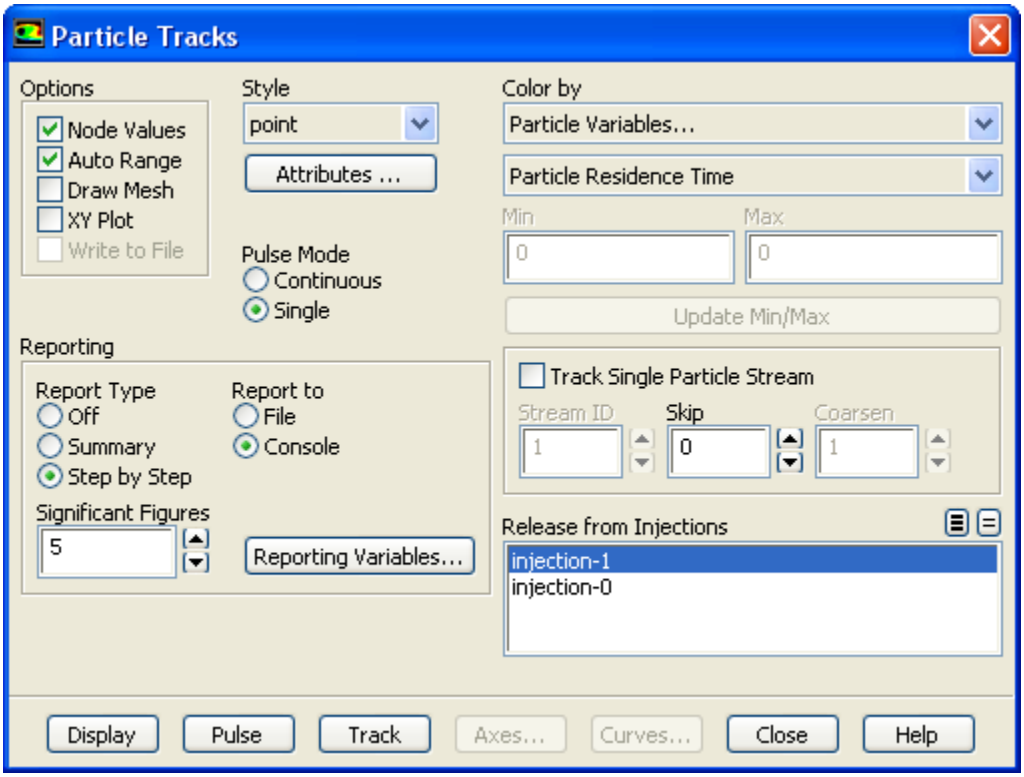
25.7.1. Displaying of Trajectories

When you have defined discrete phase particle injections, as described in *Setting Initial Conditions for the Discrete Phase* (p. 1022), you can display the trajectories of these discrete particles using the *Particle Tracks Dialog Box* (p. 1980) (*Figure 25.27* (p. 1069)).

Graphics and Animations → **Particle Tracks** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.27 The Particle Tracks Dialog Box



The procedure for drawing trajectories for particle injections is as follows:

1. Select the particle injection(s) you wish to track in the **Release from Injections** list. (You can choose to track a specific particle, instead, as described below.)
2. Set the length scale and the maximum number of steps in the *Discrete Phase Model Dialog Box* (p. 1723), as described in *Numerics of the Discrete Phase Model* (p. 1019).

   Models → **Discrete Phase** → **Edit...**

   If stochastic and/or cloud tracking is desired, set the related parameters in the **Set Injection Properties** dialog box, as described in *Stochastic Tracking* (p. 1044).
3. Set any of the display options described below.
4. Click the **Display** button to draw the trajectories or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

Important

For unsteady particle tracking simulations, clicking **Display** will show only the current location of the particles. Typically, you should select **point** in the **Style** drop-down list when displaying transient particle locations since individual positions will be displayed. The **Pulse** button option is not available for unsteady tracking.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.7.1.1. Specifying Individual Particles for Display

It is also possible to display the trajectory for an individual particle stream instead of for all the streams in a given injection. To do so, you will first need to determine which particle is of interest. Use the *Injections Dialog Box* (p. 2096) to list the particle streams in the desired injection, as described in *Creating and Modifying Injections* (p. 1038).

Define → **Injections...**

Note the ID numbers listed in the first column of the listing printed in the ANSYS FLUENT console. Then perform the following steps after step 1 above:

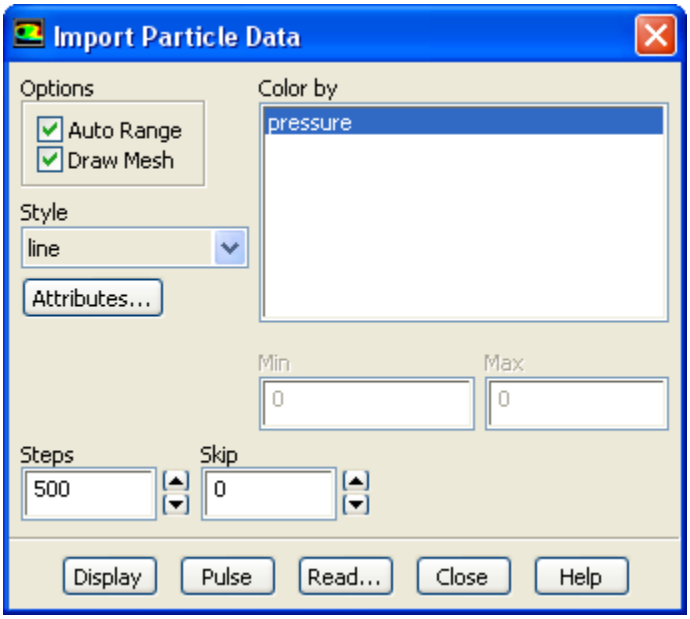
1. Enable the **Track Single Particle Stream** option in the *Particle Tracks Dialog Box* (p. 1980).
2. In the **Stream ID** field, specify the ID number of the particle stream for which you want to plot the trajectory.

25.7.1.2. Importing Particle Data

Use the **Import Particle Data** dialog box (*Figure 25.28* (p. 1070)) to import particle data to display in the graphics window.

Display → **Import Particle Data...**

Figure 25.28 The Import Particle Data Dialog Box



1. Click **Read...** to display a file selection dialog box where you can enter a file name and a directory that contains the imported data.
2. Choose from the available import options by selecting **Auto Range**and/or **Draw Mesh** under **Options**. If you prefer to restrict the range of the scalar field, disable the **Auto Range** option and set the **Min** and **Max** values manually beneath the **Color by** list.
3. Choose to color the particle pathlines by any of the scalar fields in the **Color by** list.
4. Select a pathline style under **Style**. To set pathline style attributes, click the **Attributes...** button. For more information about the pathline style types, see *Controlling the Pathline Style* (p. 1403).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The value of **Steps** sets the maximum number of steps a particle can advance. A particle will stop when it has traveled this number of steps or when it leaves the domain.
6. If your pathline plot is difficult to understand because there are too many paths displayed, you can "thin out" the pathlines by changing the **Skip** value.
7. Click the **Display** button to draw the pathlines, or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

25.7.1.3. Options for Particle Trajectory Plots

You can include the mesh in the trajectory display, control the style of the trajectories (including the twisting of ribbon-style trajectories), color them by different scalar fields and control the color scale, and coarsen trajectory plots. You can also choose node or cell values for display. If you are "pulsing" the trajectories, you can control the pulse mode. Finally, you can generate an XY plot of the particle trajectory data (e.g., residence time) as a function of time or path length and save this XY plot data to a file.

Plotting particle trajectories can be very time consuming, therefore, to reduce the plotting time, a coarsening factor can be used to reduce the number of points that are plotted. Providing a coarsening factor of n, will result in each nth point being plotted for a given trajectory in any cell. This coarsening factor is specified in the *Particle Tracks Dialog Box* (p. 1980), in the **Coarsen** field and is only valid for steady state cases. For example, if the coarsening factor is set to 2, then ANSYS FLUENT will plot alternate points.

Important

Note that if any particle or pathline enters a new cell, this point will always be plotted.

To reduce plotting time in transient cases, ANSYS FLUENT has available an option to skip plotting every n^{th} particle in an injection. Selecting this option is also done in the *Particle Tracks Dialog Box* (p. 1980) by specifying a nonzero integer in the **Skip** field. For example, if an individual stream is selected and the skip option is set to 1, every other particle will be plotted. If the entire injection is selected with a skip option of 1, every other particle will be plotted for all streams in the injection.

These options are controlled in exactly the same way that pathline-plotting options are controlled. See *Options for Pathline Plots* (p. 1402) for details about setting the trajectory plotting options mentioned above.

Note that in addition to coloring the trajectories by continuous phase variables, you can also color them according to the following discrete phase variables: particle time, particle velocity, particle diameter, particle density, particle mass, particle temperature, particle law number, particle time step, and particle Reynolds number. These variables are included in the **Particle Variables...** category of the **Color by** list. To display the minimum and maximum values in the domain, click the **Update Min/Max** button.

25.7.1.4. Graphical Display for Axisymmetric Geometries

For axisymmetric problems in which the particle has a nonzero circumferential velocity component, the trajectory of an individual particle is often a spiral about the centerline of rotation. ANSYS FLUENT displays the r and x components of the trajectory (but not the θ component) projected in the axisymmetric plane.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.7.2. Reporting of Trajectory Fates

When you perform trajectory calculations by displaying the trajectories (as described in *Displaying of Trajectories* (p. 1068)), ANSYS FLUENT will provide information about the trajectories as they are completed. By default, the number of trajectories with each possible fate (escaped, aborted, evaporated, etc.) is reported:

```
DPM Iteration ....
 num. tracked = 7, escaped = 4, aborted = 0, trapped = 0, evaporated = 3, inco
Done.
```

You can also track particles through the domain without displaying the trajectories by clicking the **Track** button at the bottom of the dialog box. This allows the listing of reports without also displaying the tracks.

25.7.2.1. Trajectory Fates

The possible fates for a particle trajectory are as follows:

- "Escaped" trajectories are those that terminate at a flow boundary for which the "escape" condition is set.
- "Incomplete" trajectories are those that were terminated when the maximum allowed number of time steps—as defined by the **Max. Number of Steps** input in the *Discrete Phase Model Dialog Box* (p. 1723) (see *Numerics of the Discrete Phase Model* (p. 1019)) — was exceeded.
- "Incomplete_parallel" may appear as an additional fate for parallel simulations. This means that the number of particle exchanges between partitions has been exceeded. Any remaining particles on the compute nodes are stopped which is indicated by the number following this fate. Therefore no further source terms from these particles are considered. The number of particle exchanges is limited to avoid very long computational time due to incomplete particles. You can change the default value of 1000 to a value of 20000 with a scheme command. Please contact the technical support engineer for this information.
- "Trapped" trajectories are those that terminate at a flow boundary where the "trap" condition has been set.
- "Evaporated" trajectories include those trajectories along which the particles were evaporated within the domain.
- "Aborted" trajectories are those that fail to complete due to roundoff reasons. You may want to retry the calculation with a modified length scale and/or different initial conditions.
- "Shed" trajectories are newly generated particles during the breakup of a larger droplet. They appear only if a breakup model is enabled.
- "Coalesced" trajectories are removed particles which have coalesced after particle-particle collisions. They appear only if the coalescence model is enabled.
- "Splashed" trajectories are particles which are newly generated when a particle touches a wall-film. Those trajectories appear only if the wall-film model is enabled.

25.7.2.2. Summary Reports

You can request additional detail about the trajectory fates as the particles exit the domain, including the mass flow rates through each boundary zone, mass flow rate of evaporated droplets, and composition of the particles.

1. Follow steps 1 and 2 in *Displaying of Trajectories* (p. 1068) for displaying trajectories.
2. Select **Summary** as the **Report Type** and click **Display** or **Track**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

For steady-state simulations, DPM summary data is not stored in the.`dat` file, since it is possible to track particles on single or combinations of injections. Transient simulations store this data since it is accumulated over time.

A detailed report similar to the following example will appear in the console window. (You may also choose to write this report to a file by selecting **File** as the **Report to** option, clicking the **Write...** button (which was originally the **Display** button), and specifying a file name for the summary report file in *The **Select File** Dialog Box* (p. 33).)

```
 number  tracked  =  10,  escaped  =  3,  aborted  =  0,  trapped  =  5,  evaporated  =  2,

 Fate             Number                    Elapsed  Time  (s)
                                     Min            Max            Avg        Std  Dev
 ----             ------     ----------     ----------     ----------      ----------
Evaporated             2     1.770e-003     1.114e-002     6.456e-003      4.686e-003
Escaped  -  Zone  6    3     6.043e-001     7.037e-001     6.471e-001      4.172e-002
Trapped  -  Zone  7    5     8.486e-003     1.767e-001     5.030e-002      6.421e-002

               (*)-  Mass  Transfer  Summary  -(*)

 Fate                        Mass  Flow  (kg/s)
                           Initial          Final          Change
 ----                    ----------     ----------      ----------
 Evaporated              8.333e-002     0.000e+000     -8.333e-002
 Escaped  -  Zone  6     1.167e-001     5.144e-002     -6.523e-002
 Trapped  -  Zone  7     2.000e-001     2.400e-002     -1.760e-001
 ----                    ----------     ----------      ----------
 Net                     4.000e-001     7.544e-002     -3.246e-001

                 (*)-  Energy  Transfer  Summary  -(*)

 Fate                        Heat  Rate  (W)              Change  of  Heat  (W)
                           Initial          Final        Sensible        Latent      React
 ----                    ----------     ----------     ----------     ----------   -------
 Evaporated              -3.180e+004     0.000e+000    -3.382e+002     3.214e+004     1.107
 Escaped  -  Zone  6      5.272e+005     6.519e+005    -3.487e+003     1.282e+005     1.523
 Trapped  -  Zone  7      4.954e+005     6.993e+005    -1.173e+003     2.051e+005     1.737
 ----                    ----------     ----------     ----------     ----------    ------
 Net                      9.908e+005     1.351e+006    -4.998e+003     3.654e+005     4.367

                (*)-  Combusting  Particles  -(*)

 Fate                Volatile  Content  (kg/s)             Char  Content  (kg/s)
                            Initial          Final      %Conv      Initial         Final
 ----                    ----------     ----------    -------   ----------    ----------
 Evaporated              0.000e+000     0.000e+000       0.00   0.000e+000    0.000e+000
 Escaped  -  Zone  6     9.333e-003     9.333e-003       0.00   2.133e-002    2.133e-002
 Trapped  -  Zone  7     9.333e-003     7.485e-010     100.00   2.133e-002    2.133e-002
 ----                    ----------     ----------    -------   ----------    ----------
 Net                     1.867e-002     9.333e-003      50.00   4.267e-002    4.267e-002

                (*)  -  Multicomponent  Droplet  -(*)

 Fate                            Species             Species  Content  (kg/s)
                                   Names          Initial          Final       %Conv
 ----                            -------       ----------     ----------     -------
 Evaporated              c5h12-droplet<l>      1.667e-002     0.000e+000      100.00
 Evaporated              c7h16-droplet<l>      3.333e-002     0.000e+000      100.00
 Evaporated                        h2o<l>      0.000e+000     0.000e+000        0.00
 Escaped  -  Zone  6     c5h12-droplet<l>      1.667e-002     2.585e-004       98.45
 Escaped  -  Zone  6     c7h16-droplet<l>      0.000e+000     0.000e+000        0.00
 Escaped  -  Zone  6               h2o<l>      3.333e-002     1.134e-002       65.99
 Trapped  -  Zone  7     c5h12-droplet<l>      3.333e-002     0.000e+000      100.00
 Trapped  -  Zone  7     c7h16-droplet<l>      3.333e-002     0.000e+000      100.00
 Trapped  -  Zone  7               h2o<l>      3.333e-002     0.000e+000      100.00
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The report groups together particles with each possible fate, and reports the number of particles, the time elapsed during trajectories, and the mass and energy transfer. This information can be very useful for obtaining information such as where particles are escaping from the domain, where particles are colliding with surfaces, and the extent of heat and mass transfer to/from the particles within the domain. Additional information is reported for combusting particles and multicomponent particles.

25.7.2.2.1. Elapsed Time

The number of particles with each fate is listed under the `Number` heading. (Particles that escape through different zones or are trapped at different zones are considered to have different fates, and are therefore listed separately.) The minimum, maximum, and average time elapsed during the trajectories of these particles, as well as the standard deviation about the average time, are listed in the `Min`, `Max`, `Avg`, and `Std Dev` columns. This information indicates how much time the particle(s) spent in the domain before they escaped, aborted, evaporated, or were trapped.

```
Fate          Number                          Elapsed  Time  (s)
                                     Min           Max            Avg       Std  Dev
----                ------      ----------     ----------    ----------      ----------   ---
Incomplete              2       1.485e+01      2.410e+01     1.947e+01       4.623e+00
Escaped  -  Zone  7     8       4.940e+00      2.196e+01     1.226e+01       4.871e+00
```

Also, on the right side of the report are listed the injection name and index of the trajectories with the minimum and maximum elapsed times. (You may need to use the scroll bar to view this information.)

```
  Elapsed  Time  (s)                                Injection,  Index
Min            Max           Avg      Std  Dev                      Min                 Max
---     ----------    ----------    ----------    --------------------  ------------------
+01      2.410e+01     1.947e+01     4.623e+00        injection-0    1    injection-0    0
+00      2.196e+01     1.226e+01     4.871e+00        injection-0    9    injection-0    2
```

25.7.2.2.2. Mass Transfer Summary

For all droplet or combusting particles with each fate, the total initial and final mass flow rates and the change in mass flow rate are reported in the `Initial`, `Final`, and `Change` columns. With this information, you can determine how much mass was transferred to the continuous phase from the particles.

For unsteady tracking, the report lists the time-integrated mass flow rate of the particle streams that have reached a particular fate at the current flow time. In other words, the report does not include particles that are still being tracked in the domain.

```
       (*)-  Mass  Transfer  Summary  -(*)

Fate                         Mass  Flow  (kg/s)
                         Initial          Final         Change
----                  ----------     ----------     ----------
Incomplete             1.388e-03      1.943e-04      1.194e-03
Escaped  -  Zone  7    1.502e-03      2.481e-04     -1.254e-03
```

25.7.2.2.3. Energy Transfer Summary

This report tells you how much heat was transferred from the particles to the continuous phase. The report is organized in two sections. For steady simulations, there is a `Heat Rate` and a `Change of Heat` section. For unsteady particle tracking, there is an `Energy` and a `Change of Energy` section. The `Heat Rate` and `Energy` sections are the same for all particle types, while the other sections report the change of heat due to the various transfer processes, which differ for each particle type. For steady simulations, the report lists the rate and the change of heat for the particle streams organized according to the particle stream fates. For unsteady tracking, the report lists the time integrated heat rate and change of the particle streams that have reached a particular fate at the current flow time. Note that the report does not include particles that are still being tracked in the domain.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.7.2.2.4. Heat Rate and Energy Reporting

For all particles with each fate, the total initial and final heat content are reported in the `Initial` and `Final` columns. The particle heat content H_p is defined as follows:

Inert Particles:

$$H_p = m_p \int_{T_{ref}}^{T_p} C_{p_p} dT \tag{25–11}$$

where:

m_p = mass flow rate of particles (kg/s)

T_p = temperature of particles (K)

C_{p_p} = heat capacity of particles (J/kg/K)

T_{ref} = reference temperature for enthalpy (K)

Droplet Particles:

$$H_p = m_p \left[-H_{lat_{ref}} + H_{pyrol} + \int_{T_{ref}}^{T_p} C_{p_p} dT \right] \tag{25–12}$$

where:

H_{pyrol} = heat of pyrolysis (J/kg)

$H_{lat_{ref}}$ = latent heat of evaporation at reference conditions (J/kg)

The latent heat at the reference conditions $H_{lat_{ref}}$ is defined in Equation 16–302 in the Theory Guide.

Combusting Particles:

$$H_p = m_p \left[-H_{lat_{ref}} + H_{comb} + \int_{T_{ref}}^{T_p} C_{p_p} dT \right] \tag{25–13}$$

where:

H_{comb} = heat of reaction (J/kg)

$H_{lat_{ref}}$ = latent heat of devolatilization adjusted to account for the difference in heat capacities between the particle and the devolatilizing species (J/kg)

Important

The `Heat Rate` section of the report is not provided for the multiple surface reactions model.

Multicomponent Particles:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_p = m_p \sum_i \left(y_i H_{p_i} \right) \quad \textbf{(25–14)}$$

where: y_i = mass fraction of component i in particle

H_{p_i} = heat content of component i

and

$$H_{p_i} = \left[-H_{lat_{ref_i}} + H_{pyrol_i} + \int_{T_{ref}}^{T_p} C_{p_{p_i}} dT \right] \quad \textbf{(25–15)}$$

where: H_{pyrol_i} = heat of pyrolysis for component i (J/kg)

$H_{lat_{ref_i}}$ = latent heat of evaporation at reference conditions for component i (J/kg)

$C_{p_{p_i}}$ = specific heat of component i (J/kg/K)

25.7.2.2.4.1. Change of Heat and Change of Energy Reporting

This section reports the total heat transferred from the particle to the continuous phase and is analyzed in components of `Sensible` heat, `Latent` heat and heat of `Reaction`. The `Total` change reported equals the difference between the `Initial` and `Final` states of the particle streams. The sensible heat component is reported for all particle types, the latent heat for the droplet, combusting and multicomponent particle, while the heat of reaction is reported for the combusting particle type only. A positive `Change of Heat` denotes that heat is expelled from the continuous phase and absorbed by the particle, while a negative `Change of Heat` denotes heat is released by the particle to the continuous phase.

Steady and Transient Simulations

For steady simulations the report lists the heat rate H_p, while for unsteady tracking the time integrated energy E_p from time 0 to current flow time ts is reported.

$$E_p = \int_0^{ts} H_P(t)\, dt \quad \textbf{(25–16)}$$

Below is an example of an `Energy Transfer Summary` report for evaporating droplets:

```
        (*)-  Energy  Transfer  Summary  -(*)
Fate                    Heat  Rate  (W)          Change  of  Heat  (W)
                      Initial         Final       Sensible       Latent        Total
----                ----------    ----------     ----------   ----------   ----------
Evaporated         -4.530e+004    0.000e+000    -4.750e+002   4.577e+004   4.530e+004
Escaped-Zone  6    -1.723e+005   -4.670e+004    -2.552e+003   1.282e+005   1.256e+005
Trapped-Zone  7    -2.176e+005    0.000e+000    -1.058e+003   2.187e+005   2.176e+005
----                ----------    ----------     ----------   ----------   ----------
Net                -4.353e+005   -4.670e+004    -4.085e+003   3.927e+005   3.886e+005
```

Below is an example of an `Energy Transfer Summary` report for combusting particles:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      (*)-  Energy  Transfer  Summary  -(*)

  Fate                 Heat  Rate  (W)            Change  of  Heat  (W)
                      Initial         Final      Sensible        Latent       Reaction      Tot
  ----              ----------    ----------    ----------    ----------     ----------   ------
Escaped-Zone  5     1.697e+005    2.555e+004    3.166e+003    1.034e+000    -1.473e+005    -1.44
Trapped-Zone  6     1.886e+004    1.938e+004    5.731e+002    1.149e-001    -5.370e+001     5.19
  ----              ----------    ----------    ----------    ----------     ----------   ------
  Net               1.886e+005    4.493e+004    3.739e+003    1.149e+000    -1.474e+005    -1.43
```

Important

In a coupled calculation, for all types of steady flows, the `Total Net Change of Heat` reported in the `Energy Transfer Summary` should balance with the opposite of the `Sum` over all fluid cells of the `DPM Sensible Enthalpy Source`. If this is not the case, this means that the coupled discrete-continuous phase calculation has not converged, and more DPM phase iterations are required. For more information on coupled calculations, see *Performing Trajectory Calculations* (p. 1064).

```
                                Sum
   DPM  Sensible  Enthalpy  Source                        (w)
  --------------------------------       --------------------
                           fluid-1                 -388937.41
```

25.7.2.2.5. Combusting Particles

If combusting particles are present, ANSYS FLUENT will include additional reporting on the volatiles and char converted. These reports are intended to help you identify the composition of the combusting particles as they exit the computational domain.

```
                  (*)-  Combusting  Particles  -(*)
  Fate                  Volatile  Content  (kg/s)           Char  Content  (kg/s)
                            Initial          Final     %Conv       Initial        Final     %Conv
  ----                   ----------     ----------   -------    ----------    ---------   -------
  Incomplete              6.247e-04      0.000e+00    100.00     5.691e-04    0.000e+00    100.00
  Escaped  -  Zone  7     6.758e-04      0.000e+00    100.00     6.158e-04    3.782e-05     93.86
```

The total volatile content at the start and end of the trajectory is reported in the `Initial` and `Final` columns under `Volatile Content`. The percentage of volatiles that has been devolatilized is reported in the `%Conv` column.

The total reactive portion (char) at the start and end of the trajectory is reported in the `Initial` and `Final` columns under `Char Content`. The percentage of char that reacted is reported in the `%Conv` column.

25.7.2.2.6. Combusting Particles with the Multiple Surface Reaction Model

If the multiple surface reaction model is used with combusting particles, ANSYS FLUENT will include additional reporting on the mass of the individual solid species that constitute the particle mass.

```
        (*)-  Multiple  Surface  Reactions  -(*)
  Fate                  Species       Species  Content  (kg/s)
                         Names       Initial        Final     %Conv
  ----                 -------    ----------   ----------   -------
  Escaped  -  Zone  6     c<s>     6.080e-02    1.487e-06    100.00
  Escaped  -  Zone  6     s<s>     3.200e-03    5.077e-06     99.84
  Escaped  -  Zone  6      cao     0.000e+00    1.153e-03      0.00
  Escaped  -  Zone  6    caso4     0.000e+00    9.266e-04      0.00
  Escaped  -  Zone  6    caco3     8.000e-03    5.260e-03     34.25
```

The total mass of each solid species in the particles at the start and end of the trajectory is reported in the `Initial` and `Final` columns, respectively. The percentage of each species that is reacted is reported in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the %Conv column. Note that for the solid reaction products (e.g., if the mass of a solid species has increased in the particle), the conversion is reported to be 0.

25.7.2.2.7. Multicomponent Particles

If your simulation includes multicomponent particles, ANSYS FLUENT generates an additional report for the particle components.

```
                (*)  -  Multicomponent  Droplet  -(*)
 Fate                              Species            Species  Content  (kg/s)
                                    Names        Initial         Final      %Conv
 ----                              -------     ----------    ----------    -------
 Evaporated              c5h12-droplet<l>     1.667e-002    0.000e+000     100.00
 Evaporated              c7h16-droplet<l>     3.333e-002    0.000e+000     100.00
 Evaporated                        h2o<l>     0.000e+000    0.000e+000       0.00
 Escaped  -  Zone  6     c5h12-droplet<l>     1.667e-002    2.585e-004      98.45
 Escaped  -  Zone  6     c7h16-droplet<l>     0.000e+000    0.000e+000       0.00
 Escaped  -  Zone  6               h2o<l>     3.333e-002    1.134e-002      65.99
 Trapped  -  Zone  7     c5h12-droplet<l>     3.333e-002    0.000e+000     100.00
 Trapped  -  Zone  7     c7h16-droplet<l>     3.333e-002    0.000e+000     100.00
 Trapped  -  Zone  7               h2o<l>     3.333e-002    0.000e+000     100.00
```

25.7.3. Step-by-Step Reporting of Trajectories

At times, you may want to obtain a detailed, step-by-step report of the particle trajectory/trajectories. Such reports can be obtained in alphanumeric format. This capability allows you to monitor the particle position, velocity, temperature, or diameter as the trajectory proceeds.

The procedure for generating files containing step-by-step reports is listed below:

1. Follow steps 1 and 2 in *Displaying of Trajectories* (p. 1068) **Track Single Particle Stream** option.
2. Select **Step by Step** as the **Report Type**.

   Important

   This option is only available for steady-state cases. For transient cases, see *Reporting of Current Positions for Unsteady Tracking* (p. 1080).

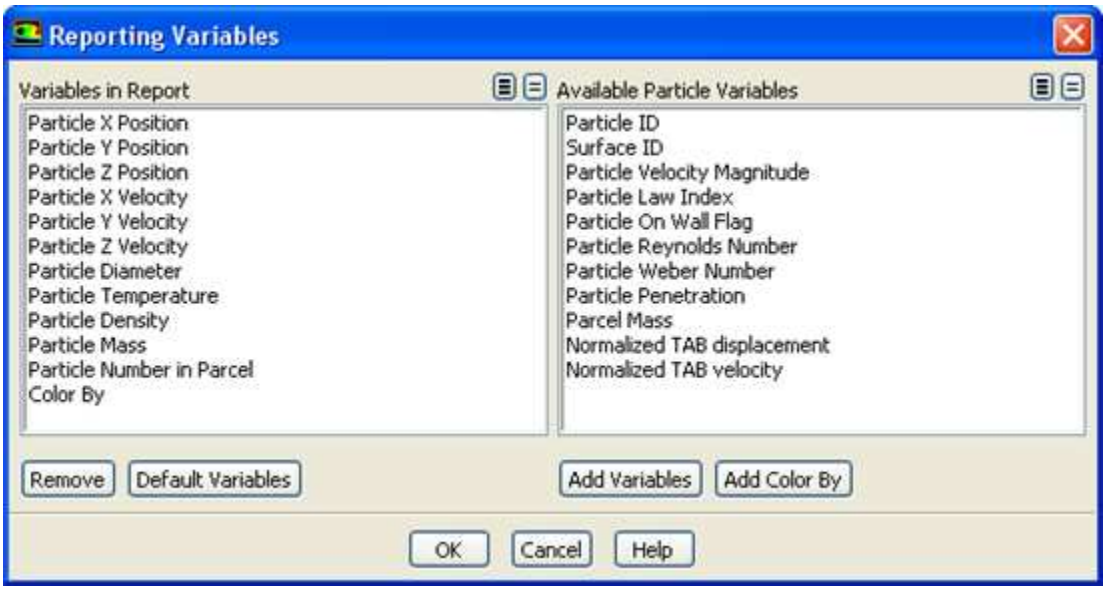
3. Select **File** as the **Report to** option. (The **Display** button will become the **Write...** button.)
4. In the **Significant Figures** field, enter the number of significant figures to be used in the step-by-step report.
5. Click the **Reporting Variables...** button. The **Reporting Variables** dialog box will appear (*Figure 25.29* (p. 1079)), where you can change the variables in the report. The list under **Variables in Report** contains all variables currently reported. The **Remove** button removes selected variables from that list. The **Default Variables** button restores the default list. The list under **Particle Variables** contains the particle variables that are available for you to select. You can add selections to the report using the **Add Variable** button. Clicking the **Add Color by** button adds the **Color by** variable to the list of **Variables in Report**, which is the only way to get cell values or customized field functions into the report.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

Figure 25.29 The Reporting Variables Dialog Box



6. Click the **Write...** button and specify a file name for the step-by-step report file in *The **Select File** Dialog Box* (p. 33).

A detailed report similar to the following example will be saved to the specified file before the trajectories are plotted. (You may also choose to print the report in the console by choosing **Console** as the **Report to** option and clicking **Display** or **Track**, but the report is very long that it is unlikely to be of use to you in that form.)

```
FILE  TYPE: 1
COLUMNS:  12
TITLE: TRACK  HISTORY

COLUMN     TYPE      VARIABLE           (UNITS)
------     ----      --------           ------
   1         2       TIME              (SECONDS)
   2        10       X-POSITION        (METERS)
   3        10       Y-POSITION        (METERS)
   4        10       Z-POSITION        (METERS)
   5        10       U-VELOCITY        (M/SEC)
   6        10       V-VELOCITY        (M/SEC)
   7        10       W-VELOCITY        (M/SEC)
   8        10       DIAMETER          (METERS)
   9        10       TEMPERATURE       (KELVIN)
  10        10       DENSITY           (KG/M3)
  11        10       MASS              (KG)
  12        10       NUMBER             -
 ---------------------------------------------

 0.00000e+00   5.00000e-02   5.00000e-02   5.00000e-02   2.00000e+01   0.0000  .  .  .
 1.07087e-07   5.00000e-02   5.00000e-02   5.00000e-02   1.23339e+01   2.6696  .  .  .
 2.51617e-07   5.00000e-02   5.00000e-02   5.00000e-02   1.04417e+01   3.3286  .  .  .
.              .             .             .             .             .
.              .             .             .             .             .
.              .             .             .             .             .
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default step-by-step report lists the position, velocity, diameter, temperature, density and mass of the particle at selected time steps along the trajectory. In addition, the variable you have selected in the **Color by** list is also included. If you choose **Console** as the **Report to** option, the variable names are written as the header of each column. (You may need to use the scroll bar to view all variables in this column.)

```
Time       X-Position   Y-Position   Z-Position    X-Velocity   Y-Velocity    Z-Veloc
0.000e+00  1.000e-03    3.120e-02    0.000e+00    1.000e+01     5.000e+00     0.000e
1.672e-05  1.168e-03    3.128e-02    0.000e+00    1.010e+01     4.988e+00     0.000e
3.342e-05  1.337e-03    3.137e-02    0.000e+00    1.019e+01     4.977e+00     0.000e
5.010e-05  1.508e-03    3.145e-02    0.000e+00    1.028e+01     4.965e+00     0.000e
6.675e-05  1.680e-03    3.153e-02    0.000e+00    1.038e+01     4.954e+00     0.000e
8.338e-05  1.854e-03    3.161e-02    0.000e+00    1.047e+01     4.942e+00     0.000e
.          .            .            .            .             .             .
.          .            .            .            .             .             .
.          .            .            .            .             .             .
```

If you change the reporting variables, only those selected will appear in the report. The particle time is always reported in the first column. Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

When the report is written to a file, a table at the beginning of the file lists all variables selected with the corresponding unit. Thus you can display or export any variable along a particle trajectory to the console or to a file.

Note that the **Coarsen** option affects the step-by-step report.

25.7.4. Reporting of Current Positions for Unsteady Tracking

In transient cases, when using unsteady tracking, you may want to obtain a report of the particle trajectory/trajectories showing the current positions of the particles. Selecting **Current Positions** under **Report Type** in the *Particle Tracks Dialog Box* (p. 1980) enables the display of the current positions of the particles.

The procedure for generating files containing current position reports is listed below:

1. Follow steps 1 and 2 in *Displaying of Trajectories* (p. 1068) for displaying trajectories. You may want to track only one particle stream at a time, using the **Track Single Particle Stream** option.
2. Select **Current Position** as the **Report Type**.
3. Select **File** as the **Report to** option. (The **Display** button will become the **Write...** button.)
4. In the **Significant Figures** field, enter the number of significant figures to be used in the step-by-step report.
5. Click the **Reporting Variables...** button. The **Reporting Variables** dialog box will appear (*Figure 25.29* (p. 1079)), where you can change the variables in the report. The list under **Variables in Report** contains all variables currently reported. The **Remove** button removes selected variables from that list. The **Default Variables** button restores the default list. The list under **Particle Variables** contains the particle variables that are available for you to select. You can add selections to the report using the **Add Variable** button. Clicking the **Add Color by** button adds the **Color by** variable to the list of **Variables in Report**, which is the only way to get cell values or customized field functions into the report.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

6. Click the **Write...** button and specify a file name for the current position report file in *The **Select File** Dialog Box* (p. 33).

The default current position report lists the position, velocity, diameter, temperature, density, mass and number in parcel of the particle at selected time steps along the trajectory. In addition, the variable you have selected in the **Color by** list is also included. If you change the reporting variables, only those selected will appear in the report. The particle time is always reported in the first column. It is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

The output to a file or to the console has the same format as the step-by-step report for steady-state cases.

```
   Time        X-Position    Y-Position    Z-Position   X-Velocity    Y-Velocity    Z-Velocity
 0.000e+00     1.000e-03     3.120e-02     0.000e+00     1.000e+01     5.000e+00     0.000e+00
 1.672e-05     1.168e-03     3.128e-02     0.000e+00     1.010e+01     4.988e+00     0.000e+00
 3.342e-05     1.337e-03     3.137e-02     0.000e+00     1.019e+01     4.977e+00     0.000e+00
 5.010e-05     1.508e-03     3.145e-02     0.000e+00     1.028e+01     4.965e+00     0.000e+00
 6.675e-05     1.680e-03     3.153e-02     0.000e+00     1.038e+01     4.954e+00     0.000e+00
 8.338e-05     1.854e-03     3.161e-02     0.000e+00     1.047e+01     4.942e+00     0.000e+00
 .             .             .             .             .             .             .
 .             .             .             .             .             .             .
 .             .             .             .             .             .             .
```

Also listed are the diameter, temperature, density, mass of the particles, number in parcel and the variable selected from the **Color by** list. (You may need to use the scroll bar to view this information.)

```
   Time        Diameter   Temperature   Density        Mass          Number     ColorBy
 9.999e-04     9.352e-05   3.710e+02   6.840e+02     2.929e-10     2.792e+02   4.783e-02
 1.999e-03     7.952e-05   3.710e+02   6.840e+02     1.801e-10     2.792e+02   3.834e-02
 3.000e-03     6.660e-05   3.710e+02   6.840e+02     1.058e-10     2.792e+02   2.989e-02
 4.001e-03     5.425e-05   3.710e+02   6.840e+02     5.719e-11     2.792e+02   3.719e-02
 5.001e-03     4.184e-05   3.710e+02   6.840e+02     2.624e-11     2.792e+02   2.978e-02
 .             .            .            .            .             .           .
 .             .            .            .            .             .           .
 .             .            .            .            .             .           .
```

25.7.5. Reporting of Interphase Exchange Terms and Discrete Phase Concentration

ANSYS FLUENT reports the magnitudes of the interphase exchange of momentum, heat, and mass in each control volume in your ANSYS FLUENT model. It can also report the total concentration of the discrete phase. You can display these variables graphically, by drawing contours, profiles, etc. They are all contained in the **Discrete Phase Model...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

- **DPM Mass Source**
- **DPM Erosion Rate**
- **DPM Accretion Rate**
- **DPM X,Y,Z Momentum Source**
- **DPM Swirl Momentum Source**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **DPM Turbulent Kinetic Energy Source**
- **DPM Turbulent Dissipation Source**
- **DPM Sensible Enthalpy Source**
- **DPM Enthalpy Source**
- **DPM Number Density**
- **DPM Volume Fraction**
- **DPM Volume**
- **DPM Number of Particles**
- **DPM Number of Parcels**
- **DPM Number of Collisions**
- **DPM Absorption Coefficient**
- **DPM Emission**
- **DPM Scattering**
- **DPM Burnout**
- **DPM Evaporation/Devolatilization**
- **DPM Concentration**
- **DPM (species) Source**
- **DPM (species) Concentration**

See *Field Function Definitions* (p. 1529) for definitions of these variables.

Note that these exchange terms are updated and displayed only when coupled calculations are performed. Displaying and reporting particle trajectories (as described in *Displaying of Trajectories* (p. 1068) and *Reporting of Trajectory Fates* (p. 1072)) will not affect the values of these exchange terms.

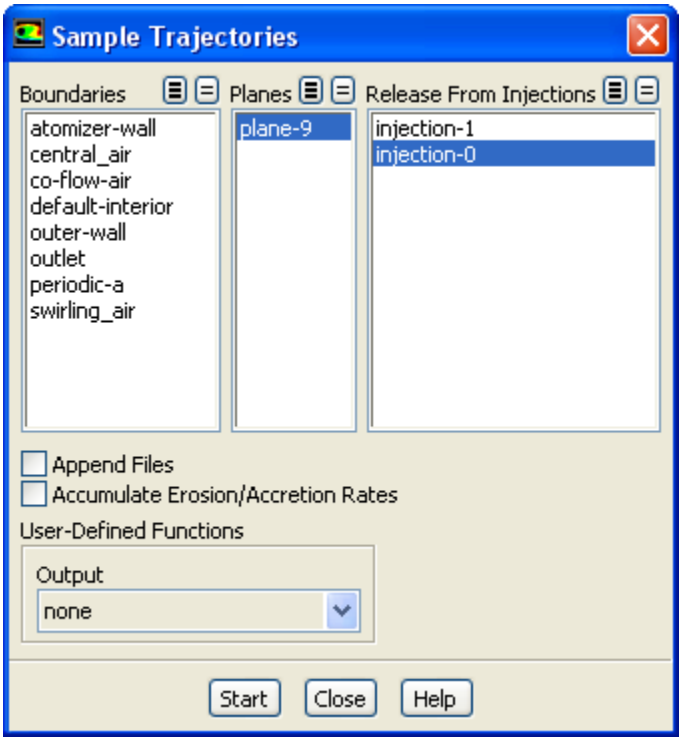
25.7.6. Sampling of Trajectories

Particle states (position, velocity, diameter, temperature, and mass flow rate) can be written to files at various boundaries and planes (lines in 2D) using the *Sample Trajectories Dialog Box* (p. 2036) (*Figure 25.30* (p. 1083)).

Reports → **Sample** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.30 The Sample Trajectories Dialog Box



The procedure for generating files containing the particle samples is listed below:

1. Select the injections to be tracked in the **Release From Injections** list.
2. Select the surfaces at which samples will be written. These can be boundaries from the **Boundaries** list or planes from the **Planes** list (in 3D) or lines from the **Lines** list (in 2D).
3. Click the **Compute** button. Note that for unsteady particle tracking, the **Compute** button will become the **Start** button (to initiate sampling) or a **Stop** button (to stop sampling).

Clicking the **Compute** button will cause the particles to be tracked and their status to be written to files when they encounter selected surfaces. The file names will be formed by appending `.dpm` to the surface name.

For unsteady particle tracking, clicking the **Start** button will open the files and write the file header sections. If the solution is advanced in time by computing some time steps, the particle trajectories will be updated and the particle states will be written to the files as they cross the selected planes or boundaries. Clicking the **Stop** button will close the files and end the sampling.

For stochastic tracking, it may be useful to repeat this process multiple times and append the results to the same file, while monitoring the sample statistics at each update. To do this, enable the **Append Files** option before repeating the calculation (clicking **Compute**). Similarly, you can cause erosion and accretion rates to be accumulated for repeated trajectory calculations by turning on the **Accumulate Erosion/Accretion Rates** option. (See also *Postprocessing of Erosion/Accretion Rates* (p. 1087).) The format and the information written for the sample output can also be controlled through a user-defined function, which can be selected in the **Output** drop-down list. More information about user-defined functions can be found in the UDF Manual.

When sampling steady particle tracks the generated sample files can be used as injection file in a file injection. Both files use a similar file format.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

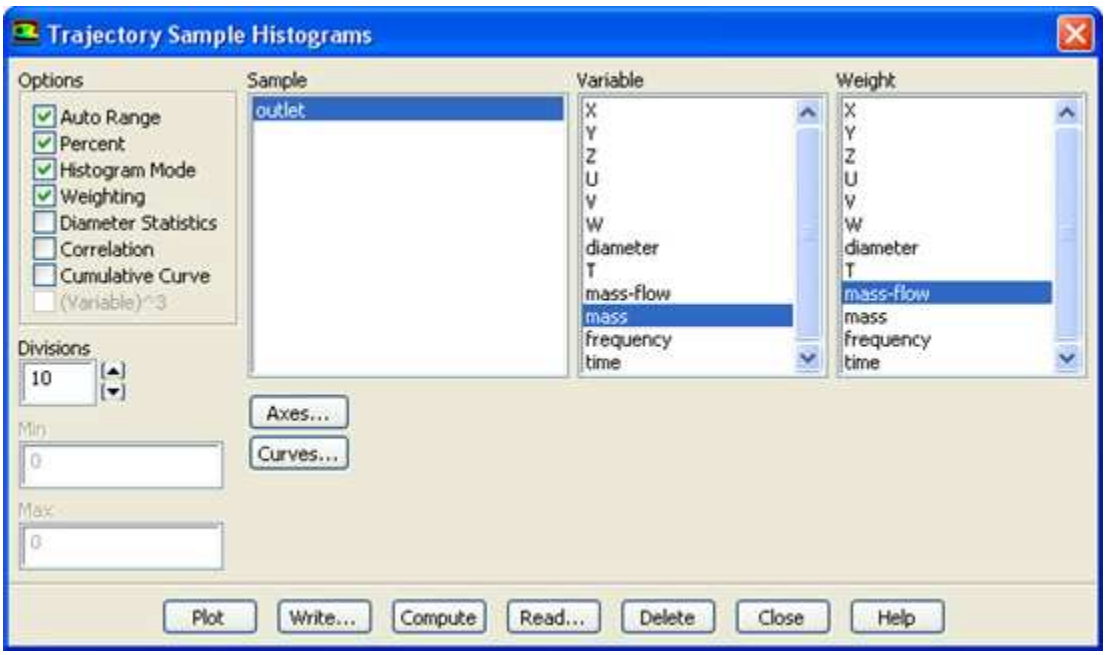
Boundedness of planes is not considered during sampling of particle tracks, which means that all particle tracks crossing the unbounded plane are sampled.

25.7.7. Histogram Reporting of Samples

Histograms can be plotted from sample files created in the *Sample Trajectories Dialog Box* (p. 2036) (as described in *Sampling of Trajectories* (p. 1082)) using the *Trajectory Sample Histograms Dialog Box* (p. 2038) (*Figure 25.31* (p. 1084)).

Reports → **Histogram** → **Set Up...**

Figure 25.31 The Trajectory Sample Histograms Dialog Box



The procedure for plotting histograms from data in a sample file is listed below:

1. Select a file to be read by clicking the **Read...** button. After you read in the sample file, the boundary name will appear in the **Sample** list.
2. Select the data sample in the **Sample** list, and then select the data to be plotted from the **Variable** list.
3. Select the data to weight the variable from the **Weight** list.
4. Click the **Plot** button at the bottom of the dialog box to display the histogram.

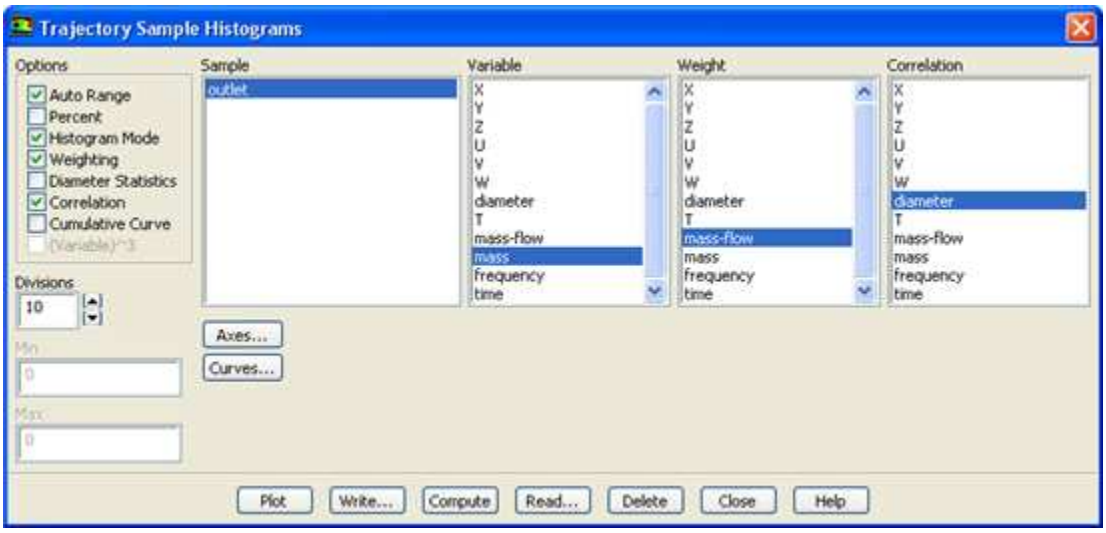
By default, the percent of particles will be plotted on the y axis. You can plot the actual number of particles by deselecting **Percent** under **Options**. The number of "bins" or intervals in the plot can be set in the **Divisions** field. You can delete samples from the list with the **Delete** button and update the **Min/Max** values with the **Compute** button. To display the histogram without the bars, deselect **Histogram Mode** under **Options**. A summary similar to that in *Summary Reporting of Current Particles* (p. 1085) will be displayed in the console for the selected variables when **Diameter Statistics** is enabled. Although these statistics are computed for the selected variable in the **Variable** list, it is applicable only to the diameter information. When the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sampled particle streams all have the same flow rate, you can disable the **Weighting**. To postprocess the histograms with other tools, you can store them in an XY-plot file format using the **Write...** button.

When investigating the behavior of particles, it is sometimes desirable to know how one type of particle variable depends on another particle variable. To facilitate this, the **Correlation** option exists. When you enable this option, an additional column of sampled variables appears, allowing you to choose the correlation variable (see *Figure 25.32* (p. 1085)).

Figure 25.32 The Trajectory Sample Histograms Dialog Box



If you want to know the continuous cumulative distributions, enable the **Cumulative Curve** option. A cumulative distribution curve is computed of the variable which is selected in the **Variable** list. However, if the **Correlation** option is enabled along with the **Cumulative Curve**, then the cumulative curve of the variable selected in the **Correlation** list is plotted. For a constant particle density, you can plot the cumulative mass distribution by selecting the diameter and enabling `(Variable)^3`.

25.7.8. Summary Reporting of Current Particles

For many mass-transfer and flow processes, it is desirable to know the mean diameter of the particles. A mean diameter, D_{jk}, is calculated from the particle size distribution using the following general expression [*42*] (p. 2211):

$$\left(D_{jk}\right)^{j-k} \equiv \frac{\int_0^\infty D^j f(D)\,dD}{\int_0^\infty D^k f(D)\,dD} \qquad \textbf{(25–17)}$$

where j and k are integers and $f(D)$ is the distribution function (e.g., Rosin-Rammler). D_{10}, for example, is the average (arithmetic) particle diameter. The Sauter mean diameter (SMD), D_{32}, is the diameter of a particle whose ratio of volume to surface area is equal to that of all particles in the computation. A summary

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of common mean diameters is given in *Table 25.6: Common Mean Diameters and Their Fields of Application* (p. 1086).

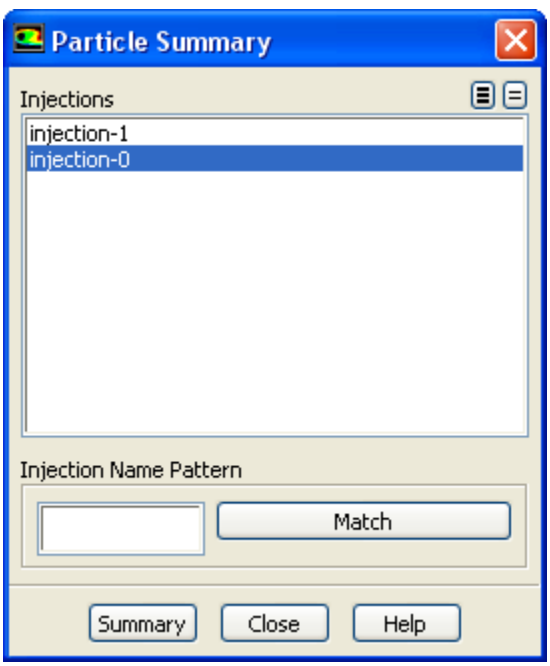
Table 25.6 Common Mean Diameters and Their Fields of Application

j	k	**Order** $j+k$	**Name**	**Field of Application**
1	0	1	Mean diameter, D_{10}	Comparisons, evaporation
2	0	2	Mean surface diameter, D_{20}	Absorption
3	0	3	Mean volume diameter, D_{30}	Hydrology
2	1	3	Overall surface diameter, D_{21}	Adsorption
3	1	4	Overall volume diameter, D_{31}	Evaporation, molecular diffusion
3	2	5	Sauter mean diameter, D_{32}	Combustion, mass transfer, and efficiency studies
4	3	7	De Brouckere diameter, D_{43}	Combustion equilibrium

Summary information (number, mass, average diameter) for particles currently in the computational domain can be reported using the *Particle Summary Dialog Box* (p. 2039) (*Figure 25.33* (p. 1086))

Reports → **Summary** → **Set Up...**

Figure 25.33 The Particle Summary Dialog Box



The procedure for reporting a summary for particle injections is as follows:

1. Select the particle injection(s) for which you want to generate a summary in the **Injections** list.

   ANSYS FLUENT provides a shortcut for selecting injections with names that match a specified pattern. To use this shortcut, enter the pattern under **Injection Name Pattern** and then click **Match** to select the injections with names that match the specified pattern. For example, if you specify `drop*`, all in-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

jections that have names beginning with **drop** (e.g., **drop-1**, **droplet**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `drop?`, all surfaces with names consisting of **drop** followed by a single character will be selected (or deselected, if they are all selected already).

2. Click **Summary** to display the injection summary in the console window.

```
 (*)-  Summary  for  Injection:  injection-0  -(*)

Total  number  of  parcels              :  1862
Total  number  of  particles            :  1.196710e+05
Total  mass                             :  1.128303e-05 (kg)
Maximum  RMS  distance  from  injector  :  7.372527e-01 (m)
Maximum  particle  diameter             :  3.072739e-04 (m)
Minimum  particle  diameter             :  1.756993e-06 (m)
Overall  RR  Spread  Parameter          :  1.446806e+00
Maximum  Error  in  RR  fit             :  1.071220e-01
Overall  RR  diameter             (D_RR):  9.051303e-05 (m)
Overall  mean  diameter           (D_10):  4.663269e-05 (m)
Overall  mean  surface  area      (D_20):  5.344694e-05 (m)
Overall  mean  volume             (D_30):  6.121478e-05 (m)
Overall  surface  diameter        (D_21):  6.125692e-05 (m)
Overall  volume  diameter         (D_31):  7.013570e-05 (m)
Overall  Sauter  diameter         (D_32):  8.030141e-05 (m)
Overall  De  Brouckere  diameter  (D_43):  1.082971e-04 (m)
```

25.7.9. Postprocessing of Erosion/Accretion Rates

You can calculate the erosion and accretion rates in a cumulative manner (over a series of injections) by using the **Sample Trajectories** dialog box. First select an injection in the **Release From Injections** list and compute its trajectory. Then enable the **Accumulate Erosion/Accretion Rates** option, select the next injection (after deselecting the first one), and click **Compute** again. The rates will accumulate at the surfaces each time you click **Compute**.

Important

Both the erosion rate and the accretion rate are defined at wall face surfaces only, so they cannot be displayed at node values.

25.8. Parallel Processing for the Discrete Phase Model

ANSYS FLUENT offers three modes of parallel processing for the discrete phase model: the **Shared Memory**, the **Message Passing**, and the **Hybrid** options under the **Parallel** tab, in the **Discrete Phase Model** dialog box. The **Shared Memory** method is suitable for computations where the machine running the ANSYS FLUENT host process is an adequately large, shared-memory, multiprocessor machine. The **Message Passing** option is suitable for generic distributed memory cluster computing. The **Hybrid** option is enabled by default and is suitable for modern multicore memory cluster computing.

Important

When tracking particles in parallel, the DPM model cannot be used with any of the multiphase flow models (VOF, mixture, or Eulerian) if the **Shared Memory** option is enabled. (Note that using the **Message Passing** or **Hybrid** option, when running in parallel, enables the compatibility of all multiphase flow models with the DPM model.)

The **Shared Memory** option (*Figure 25.34* (p. 1088)) is implemented using POSIX Threads (pthreads) based on a shared-memory model. Once the **Shared Memory** option is enabled, you can then select along with it

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **Workpile Algorithm** and specify the **Number of Threads**. By default, the **Number of Threads** is equal to the number of compute nodes specified for the parallel computation. You can modify this value based on the computational requirements of the particle calculations. If, for example, the particle calculations require more computation than the flow calculation, you can increase the **Number of Threads** (up to the number of available processors) to improve performance. When using the **Shared Memory** option, the particle calculations are entirely managed by the ANSYS FLUENT host process. You must make sure that the machine executing the host process has enough memory to accommodate the entire mesh.

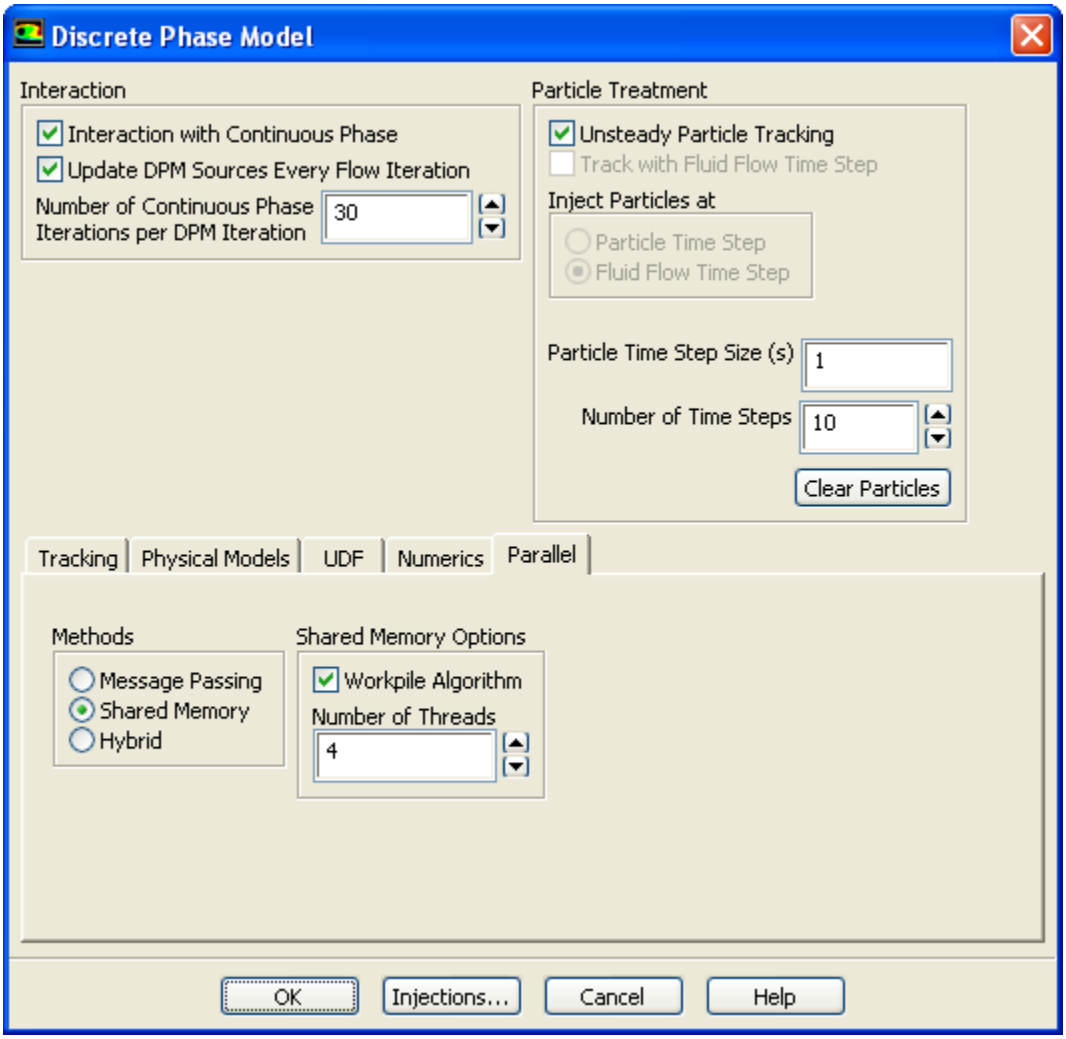
Important

Note that the **Shared Memory** option on Windows based architectures, such as ntx86 and win64 provides only serial tracking on the host, since the **Workpile Algorithm** is not available due to lack of POSIX Threads on these platforms.

Important

Please note that the **Workpile Algorithm** option is not available with the wall film boundary condition. It will be disabled automatically when choosing to simulate a wall film on a wall.

Figure 25.34 The Shared Memory Option with Workpile Algorithm Enabled



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Message Passing** option enables cluster computing and also works on shared-memory machines. With this option enabled, the compute node processes perform the particle work on their local partitions. Particle migration to other compute nodes is implemented using message passing primitives. There are no special requirements for the host machine. Note that this model is not available if the **Cloud Model** option is turned on under the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box. When running ANSYS FLUENT in parallel, by default, pathline displays are computed in serial on the host node. Pathline displays may be computed in parallel on distributed memory systems if the **Message Passing** parallel option is selected in the **Discrete Phase Model** dialog box.

The **Hybrid** option combines **Message Passing** and **OpenMP** for a dynamic load balancing without migration of cells, enables multicore cluster computing, and also works on shared-memory machines. With this option enabled, the compute node processes perform the particle calculations on their local partitions. **OpenMP** threads will be spawned, and the number of threads in each ANSYS FLUENT node process is based on the evaluation of the particle load on the current machine. The maximum number of threads on each machine can be controlled using the **Thread Control** dialog box (see *Controlling the Threads* (p. 1625) for details). The default value is the number of ANSYS FLUENT node processes on each machine. Particle migration to other compute nodes is implemented using message passing primitives. There are no special requirements for the host machine. Note that the **Hybrid** option is not available if the **Cloud Model** option is enabled in the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box. The **Message Passing** option will be used instead of the **Hybrid** option when you are only performing PDF tracking, as no load imbalance occurs in this situation. When running ANSYS FLUENT in parallel, pathline displays are computed by default in serial on the host node. Pathline displays may also be computed in parallel on distributed memory systems if the **Message Passing** option is selected from the **Methods** list.

Important

Note that the **Hybrid** option is not available for the `ntx86` platform (32–bit Windows).

You may seamlessly switch among the **Shared Memory** option, the **Message Passing** option, and the **Hybrid** option at any time during the ANSYS FLUENT session.

In addition to performing general parallel processing of the Discrete Phase Model, you have the option of implementing DPM-specific user-defined functions in parallel ANSYS FLUENT. For more information about the parallelization of DPM UDFs, see Parallelization of Discrete Phase Model (DPM) UDFs in the UDF Manual.

When using the **Message Passing** or the **Hybrid** option you can make use of ANSYS FLUENT's automated load balancing capability by giving an appropriate weight to the particle steps in each cell. In this case the number of particle steps in each partition is considered in the load balancing procedure. Further details can be found in *Using the Partitioning and Load Balancing Dialog Box* (p. 1606).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 26: Modeling Multiphase Flows

This chapter discusses the general multiphase models that are available in ANSYS FLUENT. *Introduction* (p. 1091) provides a brief introduction to multiphase modeling, *Modeling Discrete Phase* (p. 1007) discusses the Lagrangian dispersed phase model, and *Modeling Solidification and Melting* (p. 1201) describes ANSYS FLUENT's model for solidification and melting. For information about the various theories behind the general multiphase models in ANSYS FLUENT, see "Multiphase Flows" in the Theory Guide. Information about using the general multiphase models in ANSYS FLUENT is presented in the following sections:

26.1. Introduction
26.2. Steps for Using a Multiphase Model
26.3. Setting Up the VOF Model
26.4. Setting Up the Mixture Model
26.5. Setting Up the Eulerian Model
26.6. Setting Up the Wet Steam Model
26.7. Solution Strategies for Multiphase Modeling
26.8. Postprocessing for Multiphase Modeling

26.1. Introduction

The first step in solving any multiphase problem is to determine which of the regimes described in Multiphase Flow Regimes in the Theory Guide best represents your flow. Model Comparisons in the Theory Guide provides some broad guidelines for determining appropriate models for each regime, and Detailed Guidelines provides details and how to determine the degree of interphase coupling for flows involving bubbles, droplets, or particles, and the appropriate model for different amounts of coupling.

The following sections will guide you through the setup, solution, and postprocessing of multiphase flow models.

26.2. Steps for Using a Multiphase Model

The procedure for setting up and solving a general multiphase problem is outlined below, and described in detail in the subsections that follow. Remember that only the steps that are pertinent to general multiphase calculations are shown here. For information about inputs related to other models that you are using in conjunction with the multiphase model, see the appropriate sections for those models.

See also *Additional Guidelines for Eulerian Multiphase Simulations* (p. 1154) for guidelines on simplifying Eulerian multiphase simulations.

1. Enable the multiphase model you want to use (VOF, mixture, or Eulerian) and specify the number of phases. For the VOF and Eulerian models, specify the volume fraction scheme as well.

   Models → **Multiphase** → **Edit...**

   See *Enabling the Multiphase Model* (p. 1093) and *Choosing a Volume Fraction Formulation* (p. 1095) for details.

2. Copy the material representing each phase from the materials database.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials

If the material you want to use is not in the database, create a new material. See *Using the Materials Task Page* (p. 387) for details about copying from the database and creating new materials. See *Modeling Compressible Flows* (p. 1144) and *Modeling Compressible Flows* (p. 1153) for additional information about specifying material properties for a compressible phase (VOF and mixture models only). It is possible to turn off reactions in some materials by selecting **none** in the **Reactions** drop-down list under **Properties** in the **Create/Edit Materials** dialog box.

Important

If your model includes a particulate (granular) phase, you will need to create a new material for it in the *fluid* materials category (*not* the solid materials category).

3. Define the phases, and specify any interaction between them (e.g., surface tension if you are using the VOF model, slip velocity functions if you are using the mixture model, or drag functions if you are using the Eulerian model).

   Phases

   See *Defining the Phases* (p. 1098) – *Defining the Phases for the Eulerian Model* (p. 1154) for details.

4. (Eulerian model only) If the flow is turbulent, define the multiphase turbulence model.

   Models → **Viscous** → **Edit...**

   See *Modeling Turbulence* (p. 1163) for details.

5. If body forces are present, enable gravity and specify the gravitational acceleration.

   Cell Zone Conditions → **Operating Conditions...**

   See *Including Body Forces* (p. 1099) for details.

6. Specify the boundary conditions, including the secondary-phase volume fractions at flow boundaries and (if you are modeling wall adhesion in a VOF simulation) the contact angles at walls.

   Boundary Conditions

   See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

7. Set any model-specific solution parameters.

   Solution Methods

   Solution Controls

   See *Setting Time-Dependent Parameters for the VOF Model* (p. 1143) and *Solution Strategies for Multiphase Modeling* (p. 1187) for details.

8. Initialize the solution and set the initial volume fractions for the secondary phases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Initialization → **Patch...**

See *Setting Initial Volume Fractions* (p. 1189) for details.

9. Calculate a solution and examine the results. Postprocessing and reporting of results are available for each phase that is selected.

   See *Solution Strategies for Multiphase Modeling* (p. 1187) and *Postprocessing for Multiphase Modeling* (p. 1196) for details.

This section provides instructions and guidelines for using the VOF, mixture, and Eulerian multiphase models.

Information is presented in the following subsections:

26.2.1. Enabling the Multiphase Model
26.2.2. Choosing a Volume Fraction Formulation
26.2.3. Solving a Homogeneous Multiphase Flow
26.2.4. Defining the Phases
26.2.5. Including Body Forces
26.2.6. Modeling Multiphase Species Transport
26.2.7. Specifying Heterogeneous Reactions
26.2.8. Including Mass Transfer Effects
26.2.9. Defining Multiphase Cell Zone and Boundary Conditions

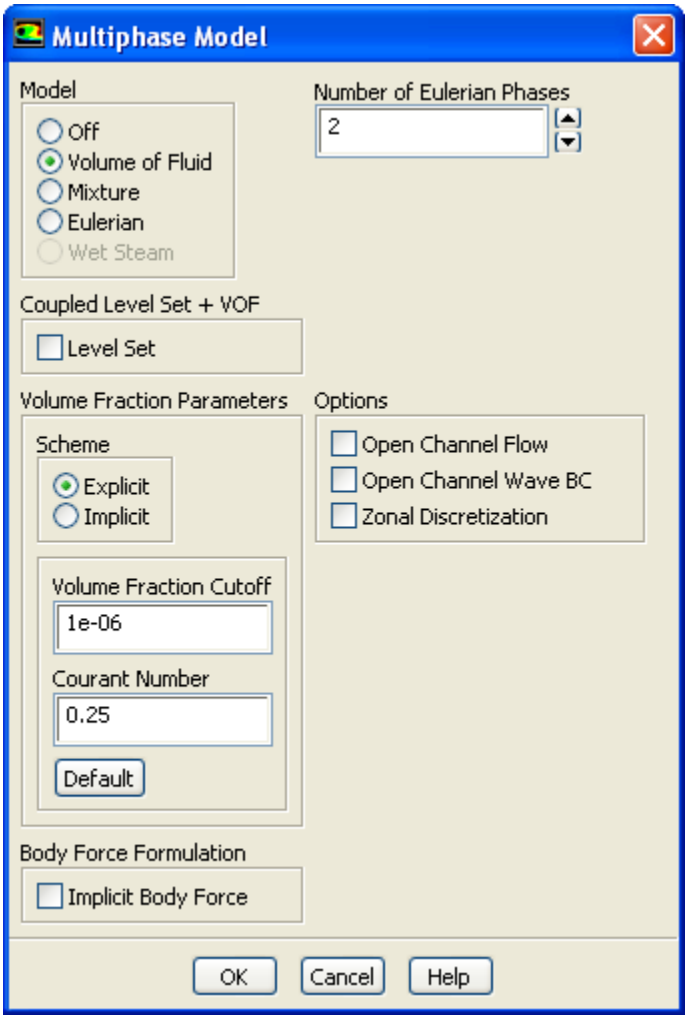
26.2.1. Enabling the Multiphase Model

To enable the VOF, mixture, or Eulerian multiphase model, select **Volume of Fluid**, **Mixture**, or **Eulerian** as the **Model** in the *Multiphase Model Dialog Box* (p. 1642) (*Figure 26.1* (p. 1094)).

Models → **Multiphase** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.1 The Multiphase Model Dialog Box



The dialog box will expand to show the relevant inputs for the selected multiphase model.

If you selected the volume of fluid (VOF) model, the inputs are as follows:

- number of phases
- (optional) coupled level set with VOF (see Coupled Level-Set and VOF Model in the Theory Guide)
- volume fraction scheme (explicit or implicit) (see *Choosing a Volume Fraction Formulation* (p. 1095))
- (optional) inclusion of open channel flow
- (optional) inclusion of open channel wave boundary conditions
- (optional) inclusion of zonal discretization for applications such as diffused interface modeling in one zone and sharp interface modeling in another zone
- (optional) inclusion of the implicit body force formulation (see *Including Body Forces* (p. 1099))

If you selected the mixture model, the inputs are as follows:

- number of phases
- (optional) inclusion of slip velocities (see *Solving a Homogeneous Multiphase Flow* (p. 1098))
- (optional) inclusion of the implicit body force formulation (see *Including Body Forces* (p. 1099))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you selected the Eulerian model, the inputs are as follows:

- number of phases
- volume fraction scheme (explicit or implicit) (see *Choosing a Volume Fraction Formulation* (p. 1095))
- (optional) including the dense discrete phase model
- (optional) including the boiling model
- (optional) including the immiscible fluid model
- (optional) including zonal discretization for applications such as diffused interface modeling in one zone and sharp interface modeling in another zone. This is only available if the **Immiscible Fluid Model** is enabled.

To specify the number of phases for the multiphase calculation, enter the appropriate value in the **Number of Eulerian Phases** field. You can specify up to 20 phases.

26.2.2. Choosing a Volume Fraction Formulation

To specify the volume fraction formulation to be used for the VOF and Eulerian multiphase models, select the appropriate **Scheme** under **Volume Fraction Parameters** in the **Multiphase Model** dialog box.

The schemes that are available in ANSYS FLUENT are **Explicit** and **Implicit**. Whether you enable the VOF or the Eulerian multiphase model, you can specify a **Volume Fraction Cutoff** value. This value is described in more detail in *Volume Fraction Limits* (p. 1097).

26.2.2.1. Explicit Schemes

Explicit schemes take on different forms:

- Time-dependent with the explicit interpolation scheme: Since the donor-acceptor scheme is available only for quadrilateral and hexahedral meshes, it cannot be used for a *hybrid* mesh containing twisted hexahedral cells. For such cases, you should use the time-dependent explicit scheme. This formulation can also be used for other cases in which the geometric reconstruction scheme does not give satisfactory results, or the flow calculation becomes unstable. Note that the CICSAM scheme, compressive scheme, or the modified HRIC scheme can be computationally inexpensive when compared to the geometric reconstruction scheme and improves the robustness and stability of the calculations. The **Volume Fraction** discretizations, **Modified HRIC**, **Compressive**, and **CICSAM**, are available in the **Solution Methods** task page when the explicit scheme is selected.

  Important

  Note that ANSYS FLUENT will automatically enable the transient formulation with first-order discretization for time in the **Solution Methods** task page.

- Time-dependent with the geometric reconstruction interpolation scheme: This formulation should be used whenever you are interested in the time-accurate transient behavior of the VOF solution.

  To use this formulation, make sure **Explicit** is selected as the **VOF Scheme** in the **Solution Methods** task page, then select **Geo-Reconstruct** as the **Volume Fraction Discretization** scheme in the **Solution Methods** task page.

- Time-dependent with the donor-acceptor interpolation scheme: This formulation should be used instead of the time-dependent formulation with the geometric reconstruction scheme if your mesh contains

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

highly twisted hexahedral cells. For such cases, the donor-acceptor scheme may provide more accurate results.

The **Donor-Acceptor** spatial discretization scheme is used when **Explicit** is selected as the **Scheme** in the **Multiphase Models** dialog box. Initially, **Donor-Acceptor** is not available in the **Solution Methods** task page GUI. To make it available, use the following text command:

`solve` → `set` → `expert`

You will be asked a series of questions, one of which is

```
Allow selection of all applicable discretization schemes? [no]
```

If your response is `yes`, then many more discretization schemes will be available for your selection. You can now use this formulation by selecting **Donor-Acceptor** as the **Volume Fraction Spatial Discretization** in the **Solution Methods** task page.

- The CICSAM scheme gives interface sharpness of the same level as the geometric reconstruction scheme and is particularly suitable for flows with high viscosity ratios between the phases.

  To use this formulation, select **Explicit** as the volume fraction **Scheme** in the *Multiphase Model Dialog Box* (p. 1642), then select **CICSAM** as the **Volume Fraction Spatial Discretization** in the **Solution Methods** task page.

  While the explicit time-dependent formulation is less computationally expensive than the geometric reconstruction scheme, the interface between phases will not be as sharp as that predicted with the geometric reconstruction scheme. To reduce this diffusivity, it is recommended that you use the second-order discretization scheme for the volume fraction equations. In addition, you may want to consider turning the geometric reconstruction scheme back on after calculating a solution with the implicit scheme, in order to obtain a sharper interface.

Important

For the geometric reconstruction and donor-acceptor schemes, if you are using a conformal mesh (i.e., if the mesh node locations are identical at the boundaries where two subdomains meet), you must ensure that there are no two-sided (zero-thickness) walls within the domain. If there are, you will need to slit them, as described in *Slitting Face Zones* (p. 193).

In general, **Geo-Reconstruct**, **Modified HRIC**, **Compressive**, and **CICSAM** are applied to cases with sharp interfaces, while **First Order Upwind** and **QUICK** are applied to diffused interfaces. Note that **Geo-Reconstruct**, **Compressive**, and **CICSAM** schemes are available with the Eulerian multiphase model, only when the **Immiscible Fluid Model** is enabled. **First Order Upwind** is available only for the Eulerian multiphase explicit scheme. It is not available when the **Immiscible Fluid Model** is enabled. However, it can be made available in the GUI for the VOF multiphase explicit scheme when the `solve/set/expert` text command is invoked.

When the **Eulerian** model is used with the **Explicit** scheme and the **Immiscible Fluid Model** is disabled, you can apply **First Order Upwind**, **QUICK**, and **Modified HRIC**. If the **Immiscible Fluid Model** is enabled, you can apply **Geo-Reconstruct**, **CICSAM**, **Compressive**, **QUICK**, and **Modified HRIC**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The issues discussed above for the explicit time-dependent formulation also apply to the implicit steady-state and time-dependent formulations, described below. You should take the precautions described above to improve the sharpness of the interface.

26.2.2.2. Implicit Schemes

Implicit schemes take on the following forms:

- Time-dependent with the implicit interpolation scheme: This formulation can be used if you are looking for a steady-state solution and you are not interested in the intermediate transient flow behavior, *but* the final steady-state solution is dependent on the initial flow conditions and/or you do not have a distinct inflow boundary for each phase.

  To use this formulation, select **Implicit** as the volume fraction **Scheme** in the *Multiphase Model Dialog Box* (p. 1642), and enable a **Transient** calculation in the **General** task page.

- Steady-state with the implicit interpolation scheme: This formulation can be used if you are looking for a steady-state solution, you are not interested in the intermediate transient flow behavior, *and* the final steady-state solution is not affected by the initial flow conditions and there is a distinct inflow boundary for each phase. Note that the implicit modified HRIC scheme can be used as a robust alternative to the explicit geometric reconstruction scheme.

  To specify the formulation when using the VOF multiphase model, select **Implicit** as the volume fraction **Scheme** in the *Multiphase Model Dialog Box* (p. 1642), then select **First Order Upwind**, **Second Order Upwind**, **Compressive**, **Modified HRIC**, **BGM** (available only for the steady state solver), or **QUICK** as the **Volume FractionSpatial Discretization** in the **Solution Methods** task page. When using the Eulerian multiphase model, select **First Order Upwind**, **QUICK**, or **Modified HRIC** as the **Volume Fraction Spatial Discretization** in the **Solution Methods** task page.

26.2.2.2.1. Examples

To help you determine the best formulation to use for your problem, some examples that use different formulations are listed below:

- jet breakup

  Use the explicit scheme (time-dependent with the geometric reconstruction scheme or the donor-acceptor if problems occur with the geometric reconstruction scheme).

- shape of the liquid interface in a centrifuge

  Use the time-dependent solver with the implicit interpolation scheme.

- flow around a ship's hull

  Use the steady-state solver with the implicit interpolation scheme.

26.2.2.3. Volume Fraction Limits

The **Volume Fraction Cutoff** allows you to specify a cutoff limit for the volume fraction values. The value that you provide is used as the lower cutoff for the volume fraction. All volume fraction values in the domain below this cutoff value are set to zero. The upper cutoff is calculated as (1.0 - lower cutoff). All volume

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fraction values above the upper cutoff value are set to 1.0. The default value is 1e-6, which is the recommended value. Using a higher value may lead to a higher volume imbalance.

Important

The **Volume Fraction Cutoff** value can be specified when using the VOF model, or when using the Eulerian Multiphase model with the **Explicit** scheme.

Note

For the **Implicit** scheme, the minimum value allowed is 0 and the maximum allowable value is 1e-6. For the **Explicit** scheme, the minimum value allowed is 0 and the maximum allowable value is 1e-4. A higher cutoff value is available with the **Explicit** scheme because it allows for the local redistribution procedure of partially filled cells, which accounts for volume loss. This treatment is not available with the **Implicit** scheme.

26.2.3. Solving a Homogeneous Multiphase Flow

If you are using the mixture model, you have the option to disable the calculation of slip velocities and solve a homogeneous multiphase flow (i.e., one in which the phases all move at the same velocity). By default, ANSYS FLUENT will compute the slip velocities for the secondary phases, as described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide. If you want to solve a homogeneous multiphase flow, turn off **Slip Velocity** under **Mixture Parameters**.

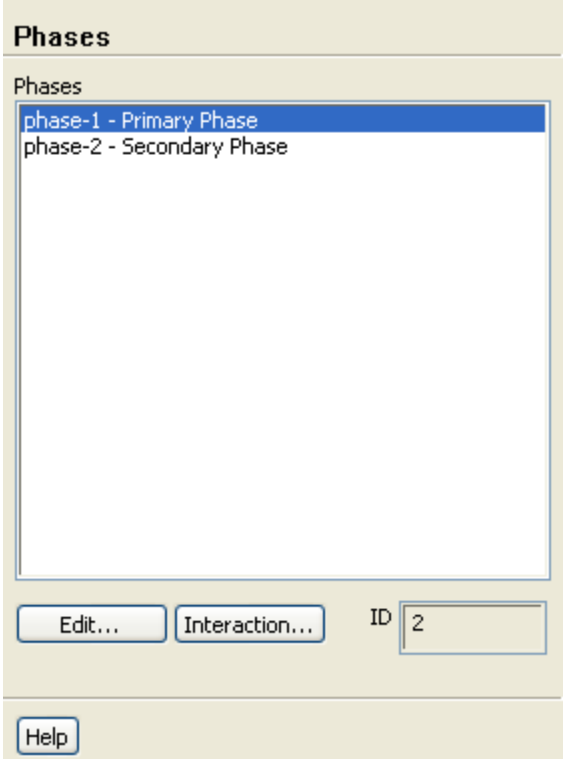
26.2.4. Defining the Phases

To define the phases (including their material properties) and any interphase interaction (e.g., surface tension and wall adhesion for the VOF model, slip velocity for the mixture model, drag functions for the mixture and the Eulerian models), (*Figure 26.2* (p. 1099)).

Phases

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.2 The Phases Task Page



Each item in the **Phases** list in this task page is one of two types: a **Primary-Phase** indicates that the selected item is the primary phase, and **Secondary-Phase** indicates that the selected item is a secondary phase. To specify any interaction between the phases, click the **Interaction...** button.

Instructions for defining the phases and interaction are provided in *Defining the Phases for the VOF Model* (p. 1136), *Defining the Phases for the Mixture Model* (p. 1145), and *Defining the Phases for the Eulerian Model* (p. 115 for the VOF, mixture, and Eulerian models, respectively.

26.2.5. Including Body Forces

When large body forces (e.g., gravity or surface tension forces) exist in multiphase flows, the body force and pressure gradient terms in the momentum equation are almost in equilibrium, with the contributions of convective and viscous terms small in comparison. Segregated algorithms converge poorly unless partial equilibrium of pressure gradient and body forces is taken into account. ANSYS FLUENT provides an optional "implicit body force" treatment that can account for this effect, making the solution more robust.

The basic procedure involves augmenting the correction equation for the face flow rate, Equation 19–49 in the Theory Guide, with an additional term involving corrections to the body force. This results in extra body force correction terms in Equation 19–47 in the Theory Guide, and allows the flow to achieve a realistic pressure field very early in the iterative process.

To include this body force, enable **Gravity** in the *Operating Conditions Dialog Box* (p. 1806) and specify the **Gravitational Acceleration**.

Cell Zone Conditions → **Operating Conditions...**

For VOF calculations, you should also enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and set the **Operating Density** to be the density of the lightest phase. (This excludes

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the buildup of hydrostatic pressure within the lightest phase, improving the round-off accuracy for the momentum balance.) If any of the phases is compressible, set the **Operating Density** to zero.

Important

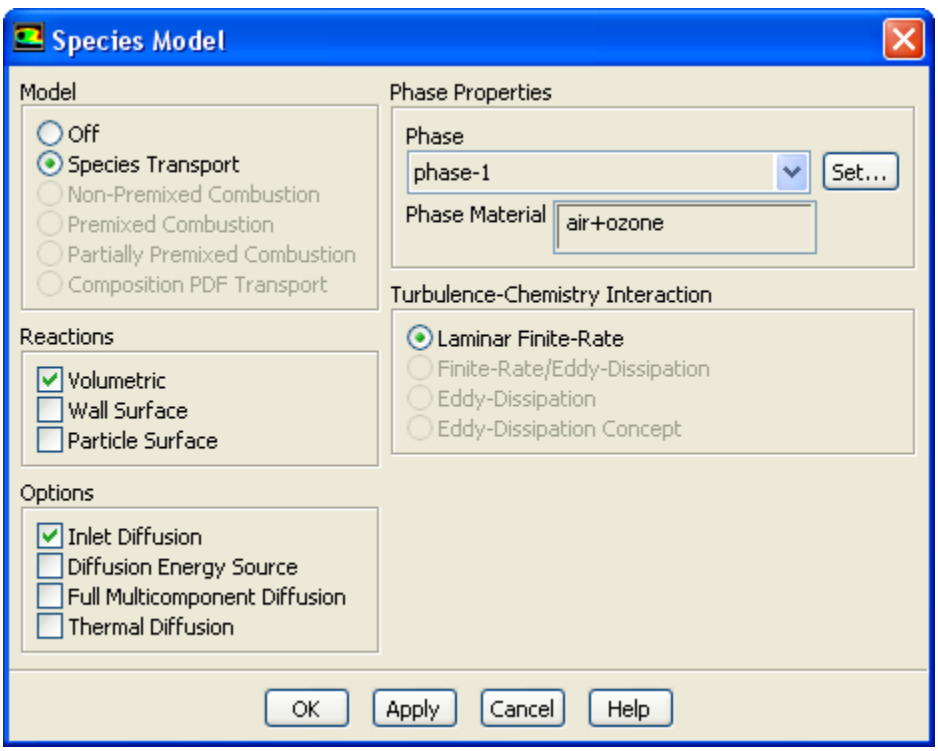
For VOF and mixture calculations involving body forces, it is recommended that you also enable the **Implicit Body Force** treatment for the **Body Force Formulation** in the **Multiphase Model** dialog box. This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and body forces in the momentum equations. See *Including Body Forces* (p. 1099) for details.

26.2.6. Modeling Multiphase Species Transport

ANSYS FLUENT lets you describe a multiphase species transport and volumetric reaction (Modeling Species Transport in Multiphase Flows in the Theory Guide) in a fashion that is similar to setting up a single-phase chemical reaction using the **Species Model** dialog box (e.g., *Figure 26.3* (p. 1100)).

Models → **Species** → **Edit...**

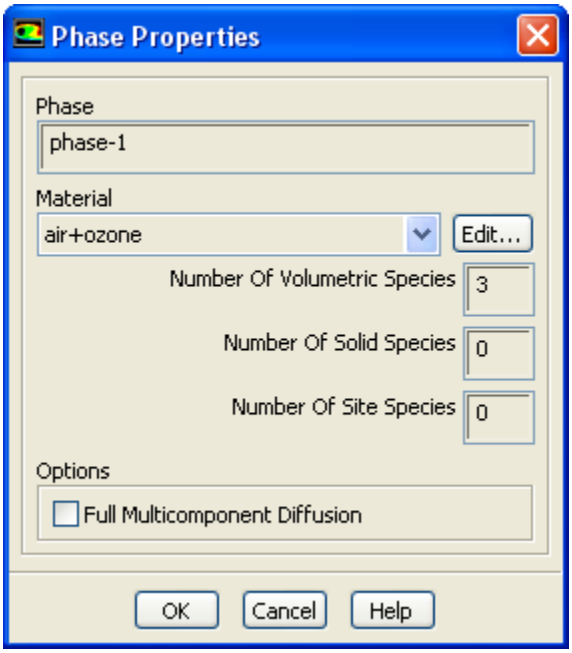
Figure 26.3 The Species Model Dialog Box with a Multiphase Model Enabled



1. Select **Species Transport** under **Model**.
2. Enable **Volumetric** under **Reactions**.
3. Select a specific phase using the **Phase** drop-down list under **Phase Properties**.
4. Click the **Set...** button to display the **Phase Properties** dialog box (*Figure 26.4* (p. 1101)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.4 The Phase Properties Dialog Box



In the **Phase Properties** dialog box, the material for each phase is listed in the **Material** drop-down list. From this list, you can choose the material that you want to use for a specific phase. The drop-down list contains all of the materials, then open the **Edit Material** dialog box by clicking the **Edit...** (or **View...**) button next to the **Material** drop-down list.

5. In the **Species Model** dialog box, choose the **Turbulence-Chemistry Interaction** model. Three models are available:

   Laminar Finite-Rate
   computes only the Arrhenius rate (see Equation 7–8 in the Theory Guide) and neglects turbulence-chemistry interaction.

   Eddy-Dissipation
   (for turbulent flows) computes only the mixing rate (see Equation 7–24 and Equation 7–25 in the Theory Guide).

   Finite-Rate/Eddy-Dissipation
   (for turbulent flows) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

When modeling multiphase species transport, additional inputs may also be required depending on your modeling needs. See, for example, *Specifying Heterogeneous Reactions* (p. 1101) for more information defining heterogeneous reactions, or *Including Mass Transfer Effects* (p. 1104) for more information on mass transfer effects.

26.2.7. Specifying Heterogeneous Reactions

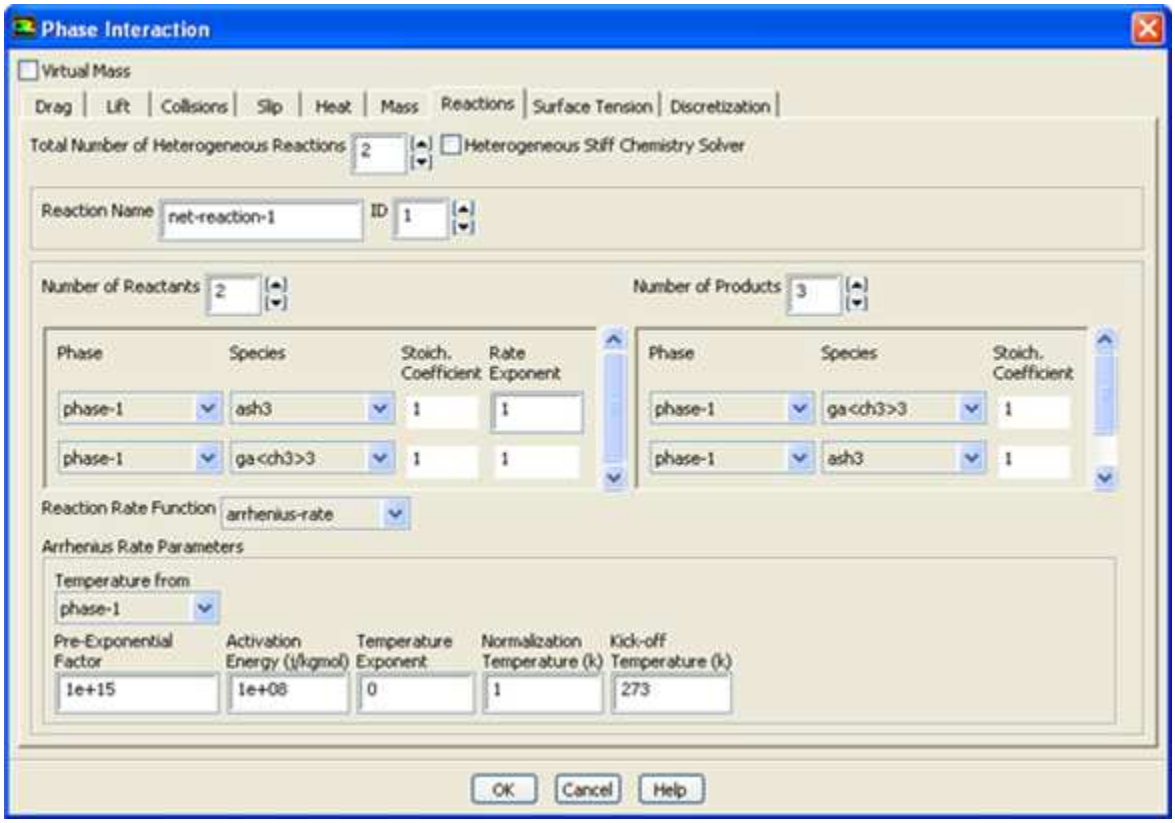
You can use ANSYS FLUENT to define multiple heterogeneous reactions and stoichiometry using the **Phase Interaction** dialog box (e.g., *Figure 26.5* (p. 1102)).

Phases → **Interaction...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Phases** task page (*Figure 26.2* (p. 1099)), click the **Interaction...** button to open the **Phase Interaction** dialog box.

Figure 26.5 The Phase Interaction Dialog Box for Heterogeneous Reactions



2. Click the **Reactions** tab in the **Phase Interaction** dialog box.
3. Set the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions) in the **Total Number of Heterogeneous Reactions** field. (Use the arrows to change the value, or type in the value and press `<Enter>`.)
4. Enable the **Heterogeneous Stiff Chemistry Solver** option if your inter-phase reaction mechanism contains numerically stiff reactions. This option can improve convergence and is available for transient Eulerian multiphase simulations. When this option is enabled, ANSYS FLUENT uses a fractional step algorithm where the flow is advanced without reaction sources for a time-step, and then the chemistry is integrated point-by-point for the same time-step. The stiff chemistry scheme solves all species in all phases coupled. Note that it is possible to include homogeneous (intra-phase) reactions along with the heterogeneous reactions in the **(Phase Interaction** dialog box (instead of in the reaction mechanism in the **Create/Edit Materials** dialog box), and these reactions will be solved with the stiff solver. The stiff ODE solver tolerances can be set using the following text command:

   `solve` → `set` → `heterogeneous-stiff-chemistry`

5. Specify the **Reaction Name** of each reaction that you want to define.
6. Set the **ID** of each reaction you want to define. (Again, if you type in the value be sure to press `Enter`.)
7. For each reaction, specify how many reactants and products are involved in the reaction by increasing the value of the **Number of Reactants** and the **Number of Products**. Select each reactant or product in the **Reaction** tab and then set its stoichiometric coefficient in the **Stoich. Coefficient** field. (The stoichiometric coefficient is the constant $v'_{i,r}$ or $v''_{i,r}$ in Equation 7–6 in the Theory Guide.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. For each reaction, indicate the **Phase** and **Species** and the stoichiometric coefficient for each of your reactants and products.
9. For each reaction, use the **Reaction Rate Function** drop-down list to select one of the following:

none
if you do not want to include a reaction rate

population-balance
is the mass transfer due to nucleation and growth. If neither the primary phase or secondary phase has species associated with it, then the mass transfer is modeled as unidirectional. If the mass transfer process involves reactions or species, the problem must be set up as one involving heterogeneous reaction/mass transfer.

Important

The **population-balance** option for **Reaction Rate Function** should not be used if there are multiple reactions leading to the formation of the secondary phase. In this case, the reaction rate functions should be specified either through the standard dialog boxes or UDFs and the growth rate function should be a sum of the individual reaction rates. This would ensure consistency between the individual reaction rates and the total mass transfer from the primary to the secondary phase.

You can always use `DEFINE_MASS_TRANSFER` or `DEFINE_HET_RXN_RATE` UDF types instead of the **population-balance** option to specify mass transfer rates. However, the growth rate function and the mass transfer rates returned from the UDFs need to be consistent with each other.

arrhenius-rate
to specify rate exponents for an Arrhenius-type reaction (see Heterogeneous Phase Interaction in the Theory Guide for more information)

Important

This simple form of the Arrhenius rate option may only be used for devolatilization reactions only. Char combustion reaction may be more involved and complicated to be simply casted in this form. Additional diffusion rate formulations may be needed to formulate a complete char (or solid phase) reaction system.

Important

Note that you can also specify the heterogeneous reaction rates using a user-defined function. A UDF is available for an Arrhenius-type reaction with rate exponents that are equivalent to the stoichiometric coefficients. For more information, see DEFINE_HET_RXN_RATE in the UDF Manual.

Important

ANSYS FLUENT assumes that the reactants are mixed thoroughly *before* reacting together, thus the heat and momentum transfer is based on this assumption. This assumption can be deactivated using a text command. For more information, contact your ANSYS FLUENT support engineer.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.8. Including Mass Transfer Effects

As discussed in Modeling Mass Transfer in Multiphase Flows in the Theory Guide, mass transfer effects in the framework of ANSYS FLUENT's general multiphase models (i.e., Eulerian multiphase, mixture multiphase, or VOF multiphase) can be modeled in one of three ways:

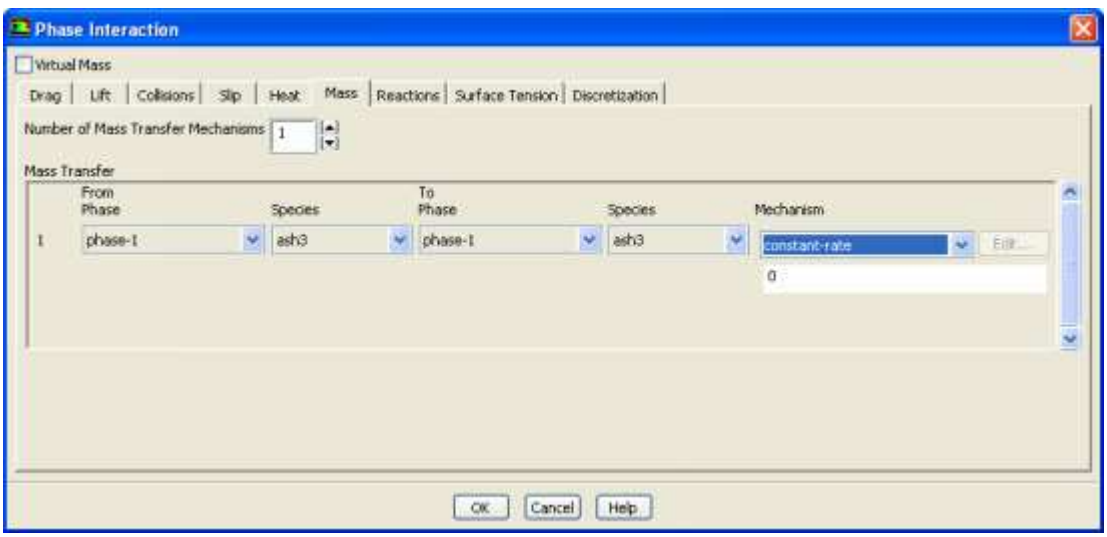
- Unidirectional constant rate mass transfer (not available for VOF calculations)
- UDF-prescribed mass transfer
- mass transfer through cavitation

Because of the different procedures and limitations involved, defining mass transfer through the Singhal et al. cavitation model is described separately in *Including Cavitation Effects* (p. 1153).

To define mass transfer in a multiphase simulation, as unidirectional constant, using a UDF, through population balance, cavitation, or evaporation and condensation, you will need to use the **Phase Interaction** dialog box (e.g., *Figure 26.6* (p. 1104)).

Phases → **Interaction...**

Figure 26.6 The Phase Interaction Dialog Box for Mass Transfer



1. Click the **Mass** tab in the **Phase Interaction** dialog box.
2. Specify the **Number of Mass Transfer Mechanisms**. You can include any number of mass transfer mechanisms in your simulation. Note also that the same pair of phases can have multiple mass transfer mechanisms and you have the ability to activate and deactivate the mechanisms of your choice.
3. For each mechanism, specify the phase of the source material under **From Phase**. Note that the phase you select for **From Phase** must be a liquid if you plan to select **cavitation**, **evaporation-condensation**, or **boiling** from the **Mechanism** drop-down menu (see Step 7.).
4. If species transport is part of the simulation, and the source phase is composed of a mixture material, then specify the species of the source phase mixture material in the corresponding **Species** drop-down list.
5. For each mechanism, specify the phase of the destination material phase under **To Phase**. Note that the phase you select for **To Phase** must be a vapor if you plan to select **cavitation**, **evaporation-condensation**, or **boiling** from the **Mechanism** drop-down menu (see Step 7.).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If species transport is part of the simulation, and the destination phase is composed of a mixture material, then specify the species of the destination phase mixture material in the corresponding **Species** drop-down list.

7. For each mass transfer mechanism, select the desired mass transfer correlation under **Mechanism**. The following choices are available:

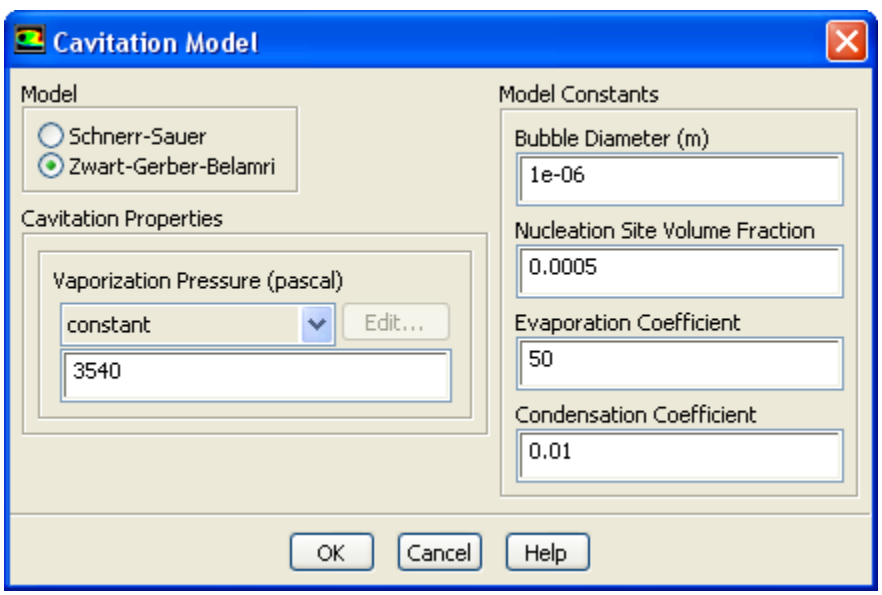
   constant-rate
   enables a constant, unidirectional mass transfer.

   user-defined
   allows you to implement a correlation reflecting a model of your choice, through a user-defined function.

   population-balance
   allows you to model flow where a number density function is introduced to account for the particle population. With the aid of particle properties (e.g., particle size, porosity, composition, etc.), different particles in the population can be distinguished and their behavior can be described. For a comprehensive understanding of this option, please refer to the Population Balance Module Manual.

   cavitation
   provides you with two model options: **Schnerr-Sauer** and **Zwart-Gerber-Belamri**. To open the **Cavitation Model** dialog box, select **cavitation** from the **Mechanism** drop-down list. For information about the cavitation models, refer to Cavitation Models in the Theory Guide.

Figure 26.7 The Cavitation Model Dialog Box



- Select **Schnerr-Sauer** and specify the **Bubble Number Density** under **Model Constants** and the **Vaporization Pressure** under **Cavitation Properties**.
- Select **Zwart-Gerber-Belamri** and specify the **BubbleDiameter**, the **Nucleation Site Volume Fraction**, the **Evaporation Coefficient**, and the **Condensation Coefficient** under **Model Constants**. Enter the **Vaporization Pressure** under **Cavitation Properties**. It is advisable to use the default values for all the model constants in both the **Schnerr-Sauer** and **Zwart-Gerber-Belamri** models. for the **Vaporization Pressure**, you have the choice of **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or **user-defined**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

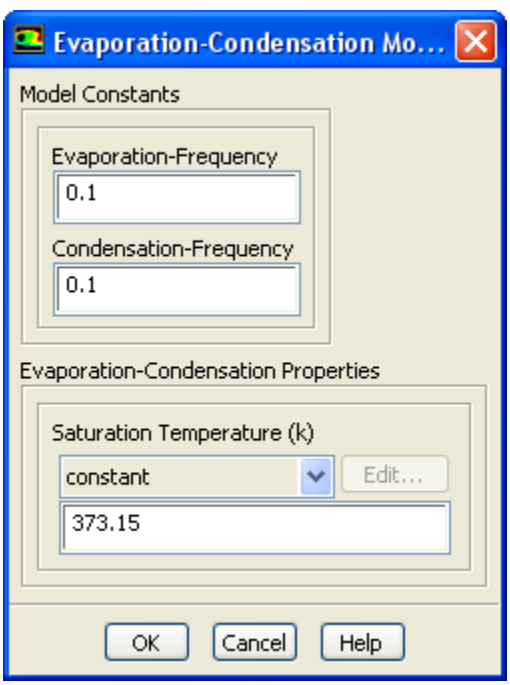
If the **Mixture** multiphase model is enabled, then the Singhal et al. cavitation model can be enabled using the `solve/set/expert` text command and responding `yes` to `use Singhal-et-al cavitation model?`. The **Singhal-Et-Al Cavitation Model** option will now be visible in the **Phase Interaction** dialog box, under the **Mass** tab. Enable this option to include the Singhal et al. cavitation model. Refer to *Including Cavitation Effects* (p. 1153) for information about setting the cavitation parameters. Also refer to Cavitation Models in the Theory Guide for information about the Singhal et al. model.

To disable this model, first deselect the **Singhal-Et-Al Cavitation Model** option in the **Phase Interaction** dialog box, then type the `solve/set/expert` text command again and enter `no` when asked if you want to `use Singhal-et-al cavitation model?`

evaporation-condensation

enables you to apply the evaporation-condensation model as the mass transfer mechanism. This model is available with the mixture and Eulerian multiphase models. Refer to Evaporation-Condensation Model in the Theory Guide for a theoretical discussion about this model.

Figure 26.8 The Evaporation-Condensation Model Dialog Box



- Enter the **Evaporation Frequency** and **Condensation Frequency** model constants. Those values are `0.1` by default. Note that the bubble diameter and accommodation coefficient are usually not very well known, which is why the coefficient $coeff$ (Equation 17–378 in the Theory Guide) can be fine tuned to match experimental data.
- Specify the **Saturation Temperature** for your flow regime.

boiling

enables you to apply the boiling model as the mass transfer mechanism. This model is only available with the Eulerian multiphase model (when **Boiling** is enabled in the **Multiphase Model** dialog

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box). For information about the inputs in *Figure 26.43* (p. 1175), please refer to *Including the Boiling Model* (p. 1171).

ANSYS FLUENT will automatically include the terms needed to model mass transfer in all relevant conservation equations. Another option to model mass transfer between phases is through the use of user-defined sources and their inclusion in the relevant conservation equations. This approach is a more involved but more powerful, allowing you to split the source terms according to a model of your choice.

Important

Momentum, energy, and turbulence are also transported with the mass that is transferred. ANSYS FLUENT assumes that the reactants are mixed thoroughly *before* reacting together, thus the heat and momentum transfer is based on this assumption. This assumption can be deactivated using a text command. For more information, contact your ANSYS FLUENT support engineer.

When your model involves the transport of multiphase species, you can define a mass transfer mechanism between species from different phases. If a particular phase does not have a specie associated with it, then the mass transfer throughout the system will be performed by the bulk fluid material.

Important

Including species transport effects in the mass transport of multiphase simulation requires that **Species Transport** be selected in the **Species Model** dialog box.

Models → **Species**

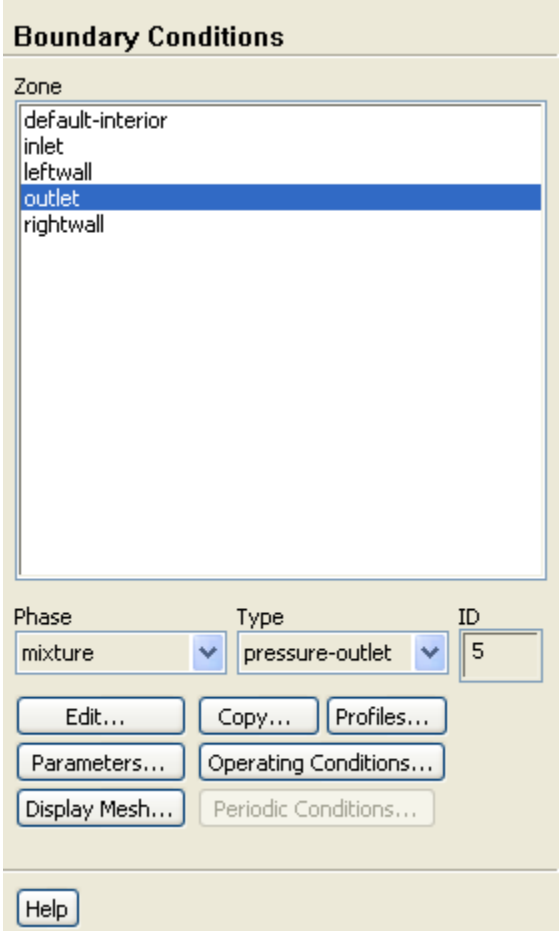
26.2.9. Defining Multiphase Cell Zone and Boundary Conditions

The procedure for setting multiphase boundary conditions is slightly different than for single-phase models. You will need to set some conditions separately for individual phases, while other conditions are shared by all phases (i.e., the mixture), as described in detail below (*Figure 26.9* (p. 1108)).

Boundary Conditions

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.9 The Boundary Conditions Task Page



26.2.9.1. Boundary Conditions for the Mixture and the Individual Phases

The conditions you need to specify for the mixture and those you need to specify for the individual phases will depend on which of the three multiphase models you are using. Details for each model are provided below.

26.2.9.1.1. VOF Model

If you are using the VOF model, the conditions you need to specify for each type of zone are listed below and summarized in *Table 26.1: Phase-Specific and Mixture Conditions for the VOF Model* (p. 1109).

- For an exhaust fan, inlet vent, intake fan, outlet vent, pressure inlet, pressure outlet, or velocity inlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to set the backflow volume fraction as a constant, a profile (see *Profiles* (p. 366)), or a user-defined function (see the UDF Manual). All other conditions are specified for the mixture.
- For a mass flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. All other conditions are specified for the mixture.

Important

Note that if you read a VOF case that was set up in a version of ANSYS FLUENT prior to 6.1, you will need to redefine the conditions at the mass flow inlets.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For an axis, fan, outflow, periodic, porous jump, radiator, solid, symmetry, or wall zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases.
- For a wall zone, you can specify the contact angle for the mixture if the wall adhesion option is enabled.
- For a fluid zone, mass sources are specified for the individual phases, and all other sources are specified for the mixture.
  - If the fluid zone is not porous, all other conditions are specified for the mixture.
  - If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See *User Inputs for Porous Media* (p. 225) for the mixture.
  - If **Zonal Discretization** was enabled in the **Multiphase Model** dialog box, then you can specify the **Compressive Scheme Slope Limiter** in the **Multiphase** tab of the **Fluid** dialog box. Depending on the value of the slope limiter, you can select either diffused or sharp interface behavior in different cell zones. The slope limiter in Equation 17–12 in the Theory Guide ranges from 0 to 2. For example, a **Compressive Scheme Slope Limiter** of 0 corresponds to first order upwind behavior, a value of 1 corresponds to second order upwind, and a value of 2 applies the compressive scheme (see The Compressive and Zonal Discretization Schemes in the Theory Guide).
  - If **Open Channel Flow** and/or **Open Channel Wave BC** is/are enabled in the **Multiphase Model** dialog box, then the **Numerical Beach** option becomes available under the **Multiphase** tab of the **Fluid** dialog box. To learn how to include numerical beach in your simulation, please refer to *Numerical Beach Treatment for Open Channels* (p. 1133).

See *Cell Zone and Boundary Conditions* (p. 205) for details about the relevant conditions for each type of boundary. Note that the pressure far-field boundary is not available with the VOF model.

Table 26.1 Phase-Specific and Mixture Conditions for the VOF Model

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; inlet vent; intake fan; outlet vent; pressure inlet; pressure outlet; velocity inlet	nothing	volume fraction	all others
mass flow inlet	mass flow/flux	mass flow/flux	all others
axis; fan; outflow; periodic; porous jump; radiator; solid; symmetry; wall	nothing	nothing	all others
pressure far-field	not available	not available	not available
fluid	mass source; other porous inputs	mass source; other porous inputs	porous zone; porosity; all others

26.2.9.1.2. Mixture Model

If you are using the mixture model, the conditions you need to specify for each type of zone are listed below and summarized in *Table 26.2: Phase-Specific and Mixture Conditions for the Mixture Model* (p. 1110).

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to set the volume fraction as a constant, a profile (see *Profiles* (p. 366)), or a user-defined function (see the UDF Manual) and if applicable, the backflow granular temperature. All other conditions are specified for the mixture.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For an inlet vent, intake fan, or pressure inlet, you will specify for the mixture which direction specification method will be used at this boundary (**Normal to Boundary** or **Direction Vector**). If you select the **Direction Vector** specification method, you will specify the coordinate system (3D only) and flow-direction components for the individual phases. For each secondary phase, you will need to set the volume fraction (as described above). All other conditions are specified for the mixture.
- For a mass flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. All other conditions are specified for the mixture.

Important

Note that if you read a mixture multiphase case that was set up in a version of ANSYS FLUENT previous to 6.1, you will need to redefine the conditions at the mass flow inlets.

- For a velocity inlet, you will specify the velocity for the individual phases. For each secondary phase, you will need to set the volume fraction (as described above). All other conditions are specified for the mixture.
- For an axis, fan, outflow, periodic, porous jump, radiator, solid, symmetry, or wall zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases. Outflow boundary conditions are not available for the cavitation model.
- For a fluid zone, mass sources are specified for the individual phases, and all other sources are specified for the mixture.

  If the fluid zone is not porous, all other conditions are specified for the mixture.

  If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See *User Inputs for Porous Media* (p. 225) for details about these inputs. All other conditions are specified for the mixture.

See *Cell Zone and Boundary Conditions* (p. 205) for details about the relevant conditions for each type of boundary. Note that the pressure far-field boundary is not available with the mixture model.

Table 26.2 Phase-Specific and Mixture Conditions for the Mixture Model

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	nothing	volume fraction	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction	coord. system; flow direction; volume fraction	dir. spec. method; all others
mass flow inlet	mass flow/flux	mass flow/flux	all others
velocity inlet	velocity	velocity; volume fraction	all others
axis; fan; outflow (n/a for cavitation model); periodic; porous jump; radiator; solid; symmetry; wall	nothing	nothing	all others
pressure far-field	not available	not available	not available
fluid	mass source; other porous inputs	mass source; other porous inputs	porous zone; porosity; all others

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.9.1.3. Eulerian Model

If you are using the Eulerian model, the conditions you need to specify for each type of zone are listed below and summarized in

- *Table 26.3: Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow)* (p. 1113)
- *Table 26.4: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model)* (p. 1114)
- *Table 26.5: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model)* (p. 1114)
- *Table 26.6: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model)* (p. 1115)

Note that the specification of turbulence parameters will depend on which of the three multiphase turbulence models you are using, as indicated in

- *Table 26.4: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model)* (p. 1114)
- *Table 26.5: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model)* (p. 1114)
- *Table 26.6: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model)* (p. 1115)

See Turbulence Models in the Theory Guide and *Modeling Turbulence* (p. 1163) for more information about multiphase turbulence models.

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase if you are modeling laminar flow or using the mixture turbulence model (the default multiphase turbulence model), except for backflow total temperature if heat transfer is on.

  For each secondary phase, you will need to set the backflow volume fraction as a constant, a profile (see *Profiles* (p. 366)), or a user-defined function (see the UDF Manual). If the phase is granular, you will also need to set its backflow granular temperature. If heat transfer is on, you will also need to set the backflow total temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.
- For an inlet vent, intake fan, or pressure inlet, you will specify for the mixture which direction specification method will be used at this boundary (**Normal to Boundary** or **Direction Vector**). If you select the **Direction Vector** specification method, you will specify the coordinate system (3D only) and flow-direction components for the individual phases. If heat transfer is on, you will also need to set the total temperature for the individual phases.

  For each secondary phase, you will need to set the volume fraction (as described above). If the phase is granular, you will also need to set its granular temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

All other conditions are specified for the mixture.

- For a mass flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. You will also need to specify the temperature of each phase, since the energy equations are solved for each phase.

  For mass flow inlet boundary conditions, you can specify the slip velocity between phases. When you select a mass flow inlet boundary for the secondary phase, two options will be available for the **Slip Velocity Specification Method**, as shown in *Figure 26.10* (p. 1112) :

  – **Velocity Ratio**

    The value for the **Phase Velocity Ratio** is the secondary phase to primary phase velocity ratio. By default, it is 1.0, which means velocities are the same (no slip). By entering a ratio that is greater than 1.0, you are indicating a larger secondary phase velocity. Otherwise, you can enter a ratio that is less than 1.0 to indicate a smaller secondary phase velocity.

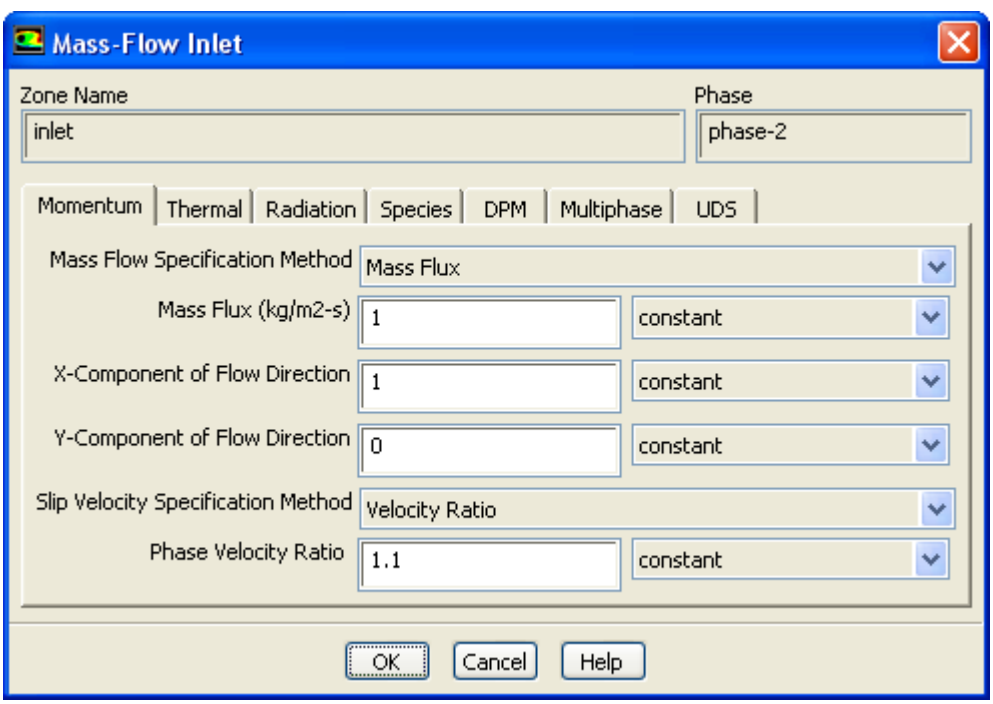
  – **Volume Fraction**

    If you specify the volume fraction at an inlet, ANSYS FLUENT will calculate the phase velocities.

    Important

    If a secondary phase has zero mass flux (i.e., the Eulerian model is used to run a single phase case), neither **Phase Velocity Ratio** nor **Volume Fraction** will affect the solution.

Figure 26.10 Mass-Flow Inlet Boundary Condition Dialog Box



- For a velocity inlet, you will specify the velocity for the individual phases. If heat transfer is on, you will also need to set the total temperature for the individual phases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each secondary phase, you will need to set the volume fraction (as described above). If the phase is granular, you will also need to set its granular temperature.

If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

All other conditions are specified for the mixture.

- For an axis, outflow, periodic, solid, or symmetry zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases.
- For a wall zone, shear conditions are specified for the individual phases. All other conditions are specified for the mixture, including thermal boundary conditions, if heat transfer is on.
- For a fluid zone, all source terms and fixed values are specified for the individual phases, *unless* you are using the mixture turbulence model or the dispersed turbulence model. If you are using the mixture turbulence model, source terms and fixed values for turbulence are specified instead for the mixture. If you are using the dispersed turbulence model, they are specified only for the primary phase.
  - If the fluid zone is not porous, all other conditions are specified for the mixture.
  - If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See *User Inputs for Porous Media* (p. 225) for details about these inputs. All other conditions are specified for the mixture.
  - If your simulation includes the **Immiscible Fluid Model** and **Zonal Discretization** was enabled in the **Multiphase Model** dialog box, then you can specify the **Compressive Scheme Slope Limiter** in the **Multiphase** tab of the **Fluid** dialog box. Depending on the value of the slope limiter, you can select either diffused or sharp interface behavior in different cell zones. The slope limiter in Equation 17–12 in the Theory Guide ranges from 0 to 2. For example, a, **Compressive Scheme Slope Limiter** of 0 corresponds to first order upwind behavior, a value of 1 corresponds to second order upwind, and a value of 2 applies the compressive scheme (see The Compressive and Zonal Discretization Schemes in the Theory Guide).

See *Cell Zone and Boundary Conditions* (p. 205) for details about the relevant conditions for each type of boundary. Note that the pressure far-field, fan, porous jump, radiator, and mass flow inlet boundaries are not available with the Eulerian model.

Table 26.3 Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	(tot. temperature)	volume fraction; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
axis; outflow; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
pressure far-field; fan; porous jump; radiator; mass flow inlet	not available	not available	not available
fluid	all source terms; all fixed values; other porous inputs	all source terms; all fixed values; other porous inputs	porous zone; porosity; all

Table 26.4 Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	(tot. temperature)	volume fraction; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
axis; outflow; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator; mass flow inlet	not available	not available	not available
fluid	other source terms; other fixed values; other porous inputs	other source terms; other fixed values; other porous inputs	source terms for turbulence; fixed values for turbulence; porous zone; porosity; all

Table 26.5 Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	turb. parameters (tot. temperature)	volume fraction; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction; turb. parameters; (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity; turb. parameters (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
axis; outflow; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
pressure far-field; fan; porous jump; radiator; mass flow inlet	not available	not available	not available
fluid	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	momentum and mass sources; momentum and mass fixed values; other porous inputs	porous zone; porosity; all

Table 26.6 Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	turb. parameters (tot. temperature)	volume fraction; turb. parameters; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction; turb. parameters (tot. temperature)	coord. system; flow direction; volume fraction; turb. parameters; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity; turb. parameters (tot. temperature)	velocity; volume fraction; turb. parameters; gran. temperature (tot. temperature)	all others
axis; outflow; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator; mass flow inlet	not available	not available	not available
fluid	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	porous zone; porosity; all others

26.2.9.2. Steps for Setting Boundary Conditions

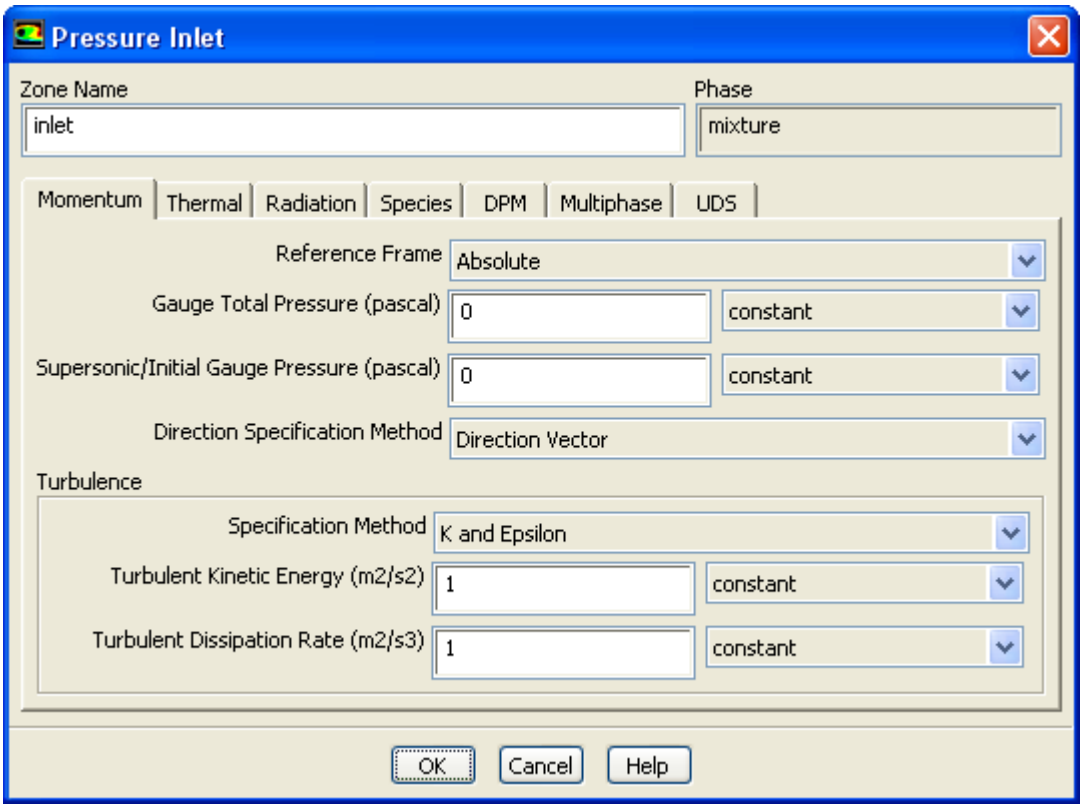
The steps you need to perform for each boundary are as follows:

1. Select the boundary in the **Zone** list in the **Boundary Conditions** task page.
2. Set the conditions for the mixture at this boundary, if necessary. (See above for information about which conditions need to be set for the mixture.)
   a. In the **Phase** drop-down list, select **mixture**.
   b. If the current **Type** for this zone is correct, click **Edit...** to open the corresponding dialog box (e.g., the **Pressure Inlet** dialog box); otherwise, choose the correct zone type in the **Type** drop-down list, confirm the change (when prompted), and the corresponding dialog box will open automatically.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. In the corresponding dialog box for the zone type you have selected (e.g., the **Pressure Inlet** dialog box for the Eulerian model, shown in *Figure 26.11* (p. 1116)), specify the mixture boundary conditions.

Figure 26.11 The Pressure Inlet Dialog Box for a Mixture



Note that only those conditions that apply to all phases, as described above, will appear in this dialog box.

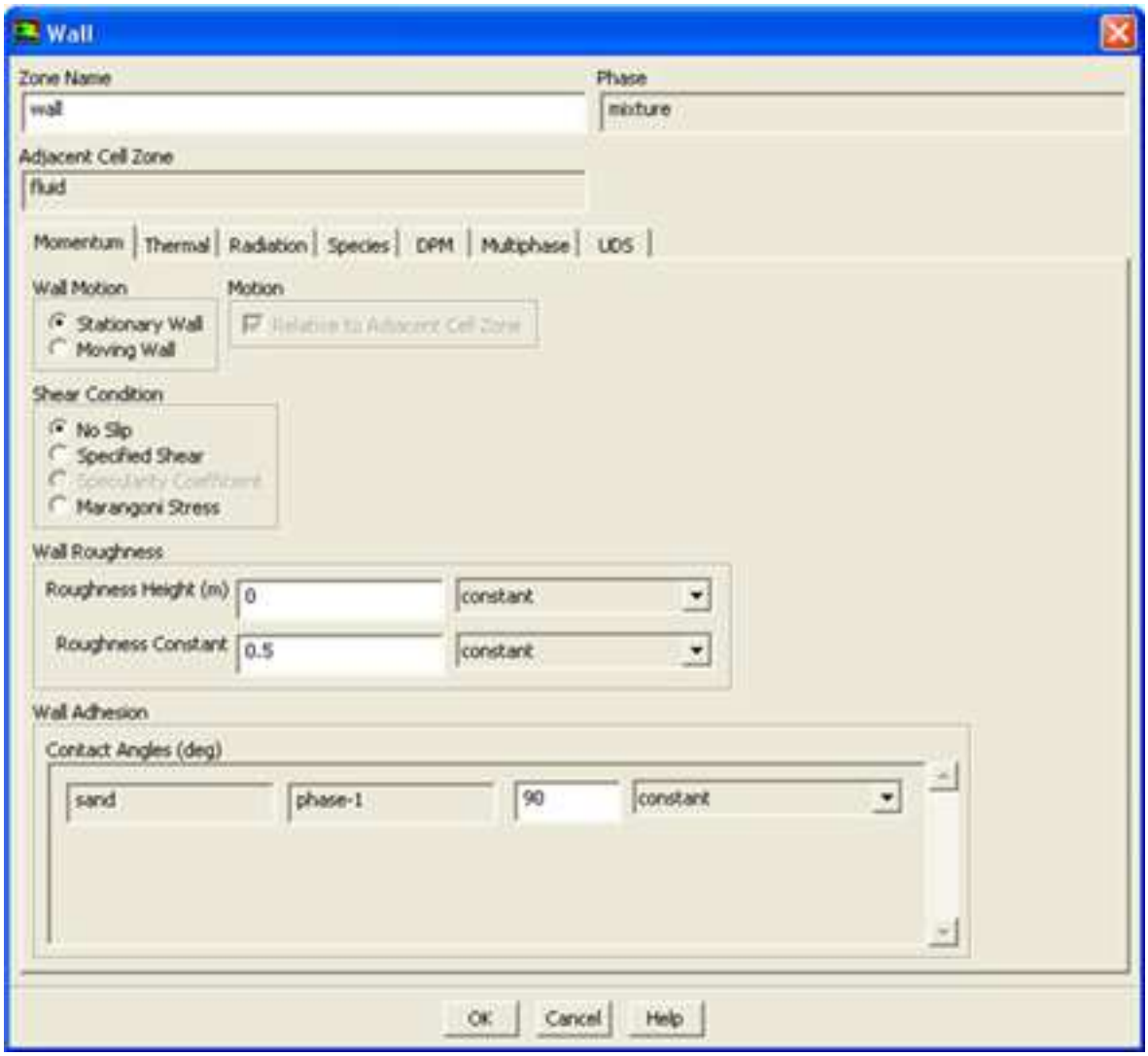
Important

For a VOF and Eulerian multiphase calculation, if you enabled the **Wall Adhesion** option in the **Phase Interaction** dialog box, you can specify the contact angle at the wall for each pair of phases as a constant (as shown in *Figure 26.12* (p. 1117)) or a UDF (see the UDF manual for more information).

The contact angle (θ_w in *Figure 26.25* (p. 1141)) is the angle between the wall and the tangent to the interface at the wall, measured inside the phase listed in the left column under **Wall Adhesion** in the **Momentum** tab of the **Wall** dialog box. For example, if you are setting the contact angle between the oil and air phases in the **Wall** dialog box shown in *Figure 26.12* (p. 1117), θ_w is measured inside the oil phase.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.12 The Wall Dialog Box for a Mixture in a VOF or Eulerian Multiphase Calculation with Wall Adhesion



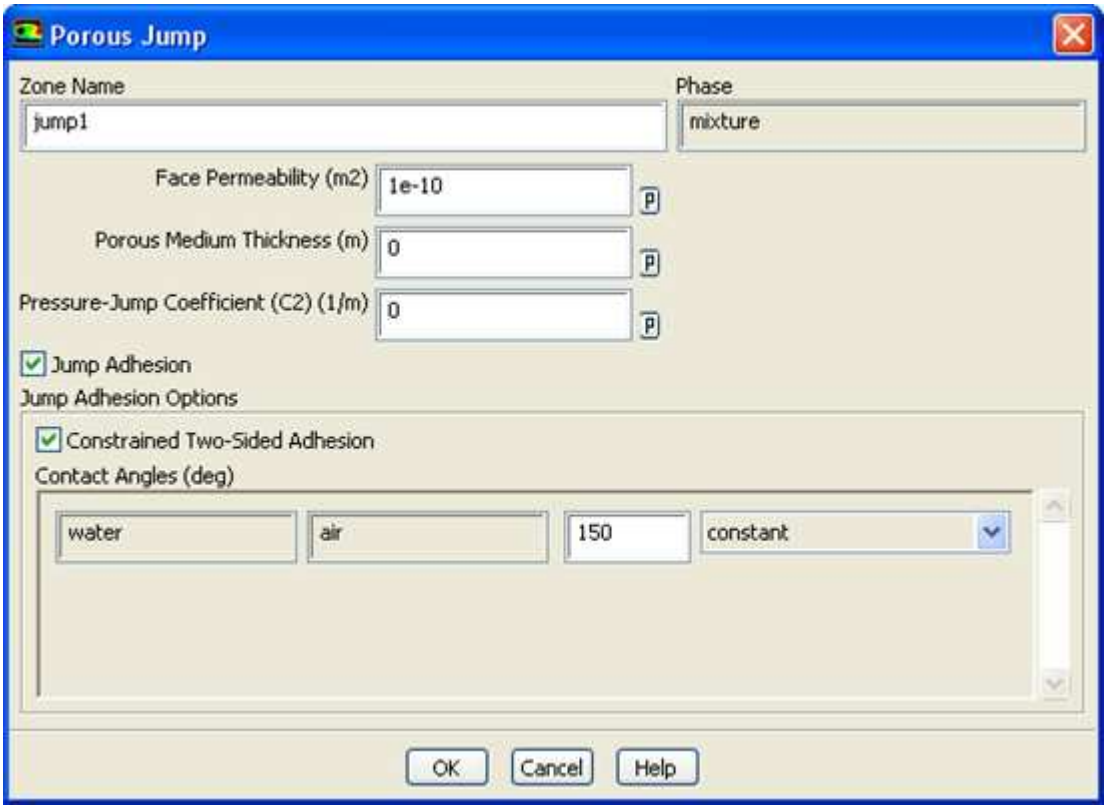
The default value for all pairs is 90 degrees, which is equivalent to no wall adhesion effects (i.e., the interface is normal to the adjacent wall). A contact angle of 45 $^{\circ}$, for example, corresponds to water creeping up the side of a container, as is common with water in a glass.

d. For the VOF model, if you enabled the **Jump Adhesion** option in the **Phase Interaction** dialog box, specify the contact angle at the porous jump for each pair of phases. If you enable the **Jump Adhesion** option in the **Phase Interaction** dialog box, this option becomes visible at each of the porous jump boundaries. You can enable or disable the **Jump Adhesion** option in the **Porous Jump** dialog boxes and provide the inputs for the contact angle at the desired porous jump boundary.

The contact angle θ_w is the angle at the porous jump. To constrain the contact angle at the porous jump based on porous or non-porous fluid zones, enable the **Constrained Two-Sided Adhesion** option. Otherwise, if it is disabled, then the forced two-sided adhesion treatment is in effect. This is described in more detail in Jump Adhesion.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.13 The Porous Jump Dialog Box Displaying Jump Adhesion



e. Click **OK** when you are done setting the mixture boundary conditions.

3. Set the conditions for each phase at this boundary, if necessary. (See above for information about which conditions need to be set for the individual phases.)

   a. In the **Phase** drop-down list, select the phase (e.g., **water**).

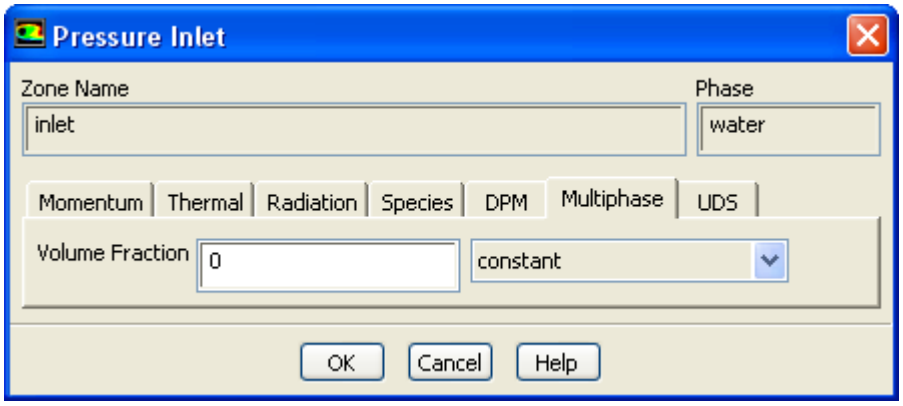
Important

Note that, when you select one of the individual phases (rather than the mixture), only one type of zone appears in the **Type** drop-down list. It is not possible to assign phase-specific zone types at a given boundary; the zone type is specified for the mixture, and it applies to all of the individual phases.

   b. Click **Edit...** to open the dialog box for this phase's conditions (e.g., the **Pressure Inlet** dialog box, shown in *Figure 26.14* (p. 1119)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.14 The Pressure Inlet Dialog Box for a Phase



c. Specify the conditions for the phase. Note that only those conditions that apply to the individual phase, as described above, will appear in this dialog box.

d. Click **OK** when you are done setting the phase-specific boundary conditions.

26.2.9.3. Steps for Copying Cell Zone and Boundary Conditions

The steps for copying cell zone and boundary conditions for a multiphase flow are slightly different from those described in *Copying Cell Zone and Boundary Conditions* (p. 208) for a single-phase flow. The modified steps are listed below:

1. In the **Cell Zone Conditions** or **Boundary Conditions** task page, click the **Copy...** button. This will open the **Copy Conditions** dialog box.
2. In the **From Cell Zone** or **From Boundary Zone** list, select the zone that has the conditions you want to copy.
3. In the **To Cell Zones** or **To Boundary Zones** list, select the zone or zones to which you want to copy the conditions.
4. In the **Phase** drop-down list, select the phase for which you want to copy the conditions (either **mixture** or one of the individual phases).

Important

Note that copying the boundary conditions for one phase does not automatically result in the boundary conditions for the other phases and the mixture being copied as well. You need to copy the conditions for each phase on each boundary of interest.

5. Click **Copy**. ANSYS FLUENT will set *all* of the selected phase's (or mixture's) boundary conditions on the zones selected in the **To Cell Zones** or **To Boundary Zones** list to be the same as that phase's conditions on the zone selected in the **From Cell Zone** or **From Boundary Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

See *Copying Cell Zone and Boundary Conditions* (p. 208) for additional information about copying boundary conditions, including limitations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3. Setting Up the VOF Model

For background information about the VOF model and the limitations that apply, refer to Overview of the VOF Model in the Theory Guide.

For additional information, please see the following sections:

26.3.1. Including Coupled Level Set with the VOF Model
26.3.2. Modeling Open Channel Flows
26.3.3. Modeling Open Channel Wave Boundary Conditions
26.3.4. Recommendations for Open Channel Initialization
26.3.5. Numerical Beach Treatment for Open Channels
26.3.6. Defining the Phases for the VOF Model
26.3.7. Setting Time-Dependent Parameters for the VOF Model
26.3.8. Modeling Compressible Flows
26.3.9. Modeling Solidification/Melting

26.3.1. Including Coupled Level Set with the VOF Model

When using the VOF formulation, you can couple the level set method with it to help overcome some limitations that exist in the interface tracking method of the VOF model and the level set method. To use the coupled level set method with VOF, perform the following:

1. Enable the volume of fluid model.
   a. Open the **Multiphase Model** dialog box.

      Models → **Multiphase** → **Edit...**

   b. Under **Model**, enable **Volume of Fluid**.
   c. Under **Coupled Level Set + VOF**, enable **Level Set** (see *Figure 26.1* (p. 1094)).

After the **Level Set** option is enabled, proceed as you normally would when setting up the VOF model (described in *Setting Up the VOF Model* (p. 1120)). For theoretical information, please refer to Coupled Level-Set and VOF Model.

Note

When using the **Level Set** option, the recommended scheme is the geo-reconstruct scheme (see The Geometric Reconstruction Scheme).

Important

- The level set method is only suitable for two-phase flow regime, where two fluids are not interpenetrating.
- The level set model can only be used when the VOF model is activated. No mass transfer is allowed.
- The level set method is not compatible with the dynamic mesh model.

Normally, zero flux of the level set function is set as the default for the boundary conditions. Due to the geometrical re-initialization procedure at each time step, the boundary conditions shall not have any signi-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ficant effect on the results. For more information, see Re-initialization of the Level-set Function via the Geometrical Method.

26.3.2. Modeling Open Channel Flows

Using the VOF formulation, open channel flows can be modeled in ANSYS FLUENT. To start using the open channel flow boundary condition, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   General

2. Enable the volume of fluid model.
   a. Open the **Multiphase Model** dialog box.

      Models → **Multiphase** → **Edit...**

   b. Under **Model**, enable **Volume of Fluid**.
   c. Under **Scheme**, select either **Implicit**, **Explicit**.
3. Under **Volume Fraction Parameters**, select **Open Channel Flow**.

Note

The default VOF formulation is set to **Implicit** after enabling the **Open Channel Flow** option. This is done to allow the usage of larger time step sizes for such applications.

In order to set specific parameters for a particular boundary for open channel flows, enable the **Open Channel** option in the **Multiphase** tab of the corresponding boundary condition dialog box. *Table 26.7: Open Channel Boundary Parameters for the VOF Model* (p. 1121) summarizes the types of boundaries available to the open channel flow boundary condition, and the additional parameters needed to model open channel flow. For more information on setting boundary condition parameters, see *Cell Zone and Boundary Conditions* (p. 205).

Table 26.7 Open Channel Boundary Parameters for the VOF Model

Boundary Type	Parameter
pressure inlet	Inlet Group ID; Secondary Phase for Inlet; Flow Specification Method; Free Surface Level, Bottom Level; Velocity Magnitude
pressure outlet	Outlet Group ID; Pressure Specification Method; Free Surface Level; Bottom Level
mass flow inlet	Inlet Group ID; Secondary Phase for Inlet; Free Surface Level; Bottom Level
outflow	Flow Rate Weighting

26.3.2.1. Defining Inlet Groups

Open channel systems involve the flowing fluid (the secondary phase) and the fluid above it (the primary phase).

If both phases enter through the separate inlets (e.g., `inlet-phase2` and `inlet-phase1`), these two inlets form an inlet group. This inlet group is recognized by the parameter **Inlet Group ID**, which will be

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

same for both the inlets that make up the inlet group. On the other hand, if both the phases enter through the same inlet (e.g., `inlet-combined`), then the inlet itself represents the inlet group.

Important

In three-phase flows, only one secondary phase is allowed to pass through one inlet group.

26.3.2.2. Defining Outlet Groups

Outlet-groups can be defined in the same manner as the inlet groups.

Important

In three-phase flows, the outlet should represent the outlet group, i.e., separate outlets for each phase are not recommended in three-phase flows.

26.3.2.3. Setting the Inlet Group

For pressure inlets and mass flow inlets, the **Inlet Group ID** is used to identify the different inlets that are part of the same inlet group. For instance, when both phases enter through the same inlet (single face zone), then those phases are part of one inlet group and you would set the **Inlet Group ID** to 1 for that inlet (or inlet group).

In the case where the same inlet group has separate inlets (different face zones) for each phase, then the **Inlet Group ID** will be the same for each inlet of that group.

When specifying the inlet group, use the following guidelines:

- Since the **Inlet Group ID** is used to identify the inlets of the same inlet group, general information such as **Free Surface Level**, **Bottom Level**, or the mass flow rate for each phase should be the same for each inlet of the same inlet group.
- You should specify a different **Inlet Group ID** for each distinct inlet group.

  For example, consider the case of two inlet groups for a particular problem. The first inlet group consists of water and air entering through the same inlet (a single face zone). In this case, you would specify an inlet group ID of 1 for that inlet (or inlet group). The second inlet group consists of oil and air entering through the same inlet group, but each uses a different inlet (`oil-inlet` and `air-inlet`) for each phase. In this case, you would specify the same **Inlet Group ID** of 2 for both of the inlets that belong to the inlet group.

26.3.2.4. Setting the Outlet Group

For pressure outlet boundaries, the **Outlet Group ID** is used to identify the different outlets that are part of the same outlet group. For instance, when both phases enter through the same outlet (single face zone), then those phases are part of one outlet group and you would set the **Outlet Group ID** to 1 for that outlet (or outlet group).

In the case where the same outlet group has separate outlets (different face zones) for each phase, then the **Outlet Group ID** will be the same for each outlet of that group.

When specifying the outlet group, use the following guidelines:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Since the **Outlet Group ID** is used to identify the outlets of the same outlet group, general information such as **Free Surface Level** or **Bottom Level** should be the same for each outlet of the same outlet group.
- You should specify a different **Outlet Group ID** for each distinct outlet group.

  For example, consider the case of two outlet groups for a particular problem. The first inlet group consists of water and air exiting from the same outlet (a single face zone). In this case, you would specify an outlet number of 1 for that outlet (or outlet group). The second outlet group consists of oil and air exiting through the same outlet group, but each uses a different outlet (`oil-outlet` and `air-outlet`) for each phase. In this case, you would specify the same **Outlet Group ID** of 2 for both of the outlets that belong to the outlet group.

Important

For three-phase flows, when all the phases are leaving through the same outlet, the outlet should consist only of a single face zone.

26.3.2.5. Determining the Free Surface Level

For the appropriate boundary, you need to specify the **Free Surface Level** value. This parameter is available for all relevant boundaries, including pressure outlet, mass flow inlet, and pressure inlet. The **Free Surface Level**, is represented by y_{local} in Equation 17–34 in the Theory Guide.

$$y_{local} = -(\vec{a} \cdot \hat{g}) \tag{26–1}$$

where $\vec{a}$ is the position vector of any point on the free surface, and $\hat{g}$ is the unit vector in the direction of the force of gravity. Here we assume a horizontal free surface that is normal to the direction of gravity.

We can simply calculate the free surface level in two steps:

1. Determine the absolute value of height from the free surface to the origin in the direction of gravity.
2. Apply the correct sign based on whether the free surface level is above or below the origin.

If the liquid's free surface level lies above the origin, then the **Free Surface Level** is positive (see *Figure 26.15* (p. 1124)). Likewise, if the liquid's free surface level lies below the origin, then the **Free Surface Level** is negative.

26.3.2.6. Determining the Bottom Level

For the appropriate boundary, you need to specify the **Bottom Level** value. This parameter is available for all relevant boundaries, including pressure outlet, mass flow inlet, and pressure inlet. The **Bottom Level**, is represented by a relation similar to Equation 17–34 in the Theory Guide.

$$y_{bottom} = -(\vec{b} \cdot \hat{g}) \tag{26–2}$$

where $\vec{b}$ is the position vector of any point on the bottom of the channel, and $\hat{g}$ is the unit vector of gravity. Here we assume a horizontal free surface that is normal to the direction of gravity.

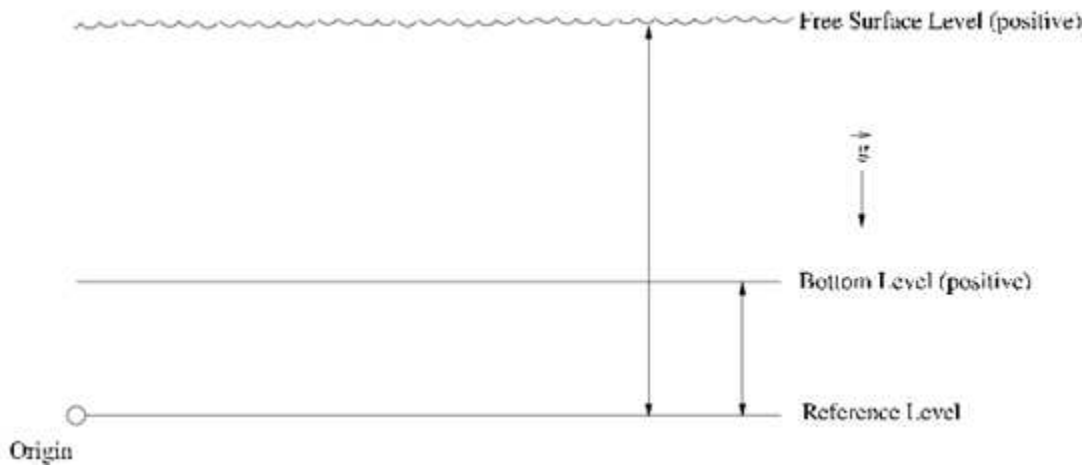
We can simply calculate the bottom level in two steps:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Determine the absolute value of depth from the bottom level to the origin in the direction of gravity.
2. Apply the correct sign based on whether the bottom level is above or below the origin.

If the channel's bottom lies above the origin, then the **Bottom Level** is positive (see *Figure 26.15* (p. 1124)). Likewise, if the channel's bottom lies below the origin, then the **Bottom Level** is negative.

Figure 26.15 Determining the Free Surface Level and the Bottom Level



26.3.2.7. Specifying the Total Height

The total height, along with the velocity, is used as an option for describing the flow. The total height is given as

$$y_{tot}=y_{local}+\frac{V^2}{2g} \qquad \textbf{(26–3)}$$

where V is the velocity magnitude and g is the gravity magnitude.

26.3.2.8. Determining the Velocity Magnitude

Pressure inlet boundaries require the **Velocity Magnitude** for calculating the dynamic pressure at the boundary. This is to be specified as the magnitude of the upstream inlet velocity in the flow.

26.3.2.9. Determining the Secondary Phase for the Inlet

For pressure inlets and mass flow inlets, the **Secondary Phase for Inlet** field is significant in cases of three-phase flows.

Important

Note that only one secondary phase is allowed to pass through one inlet group.

Consider a problem involving a three-phase flow consisting of air as the primary phase, and oil and water as the secondary phases. Consider also that there are two inlet groups:

- water and air

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- oil and air

For the former inlet group, you would choose water as the secondary phase. For the latter inlet group, you would choose oil as the secondary phase.

26.3.2.10. Choosing the Pressure Specification Method

For a pressure outlet boundary, the outlet pressure can be specified in one of three ways:

- by prescribing the local height, such as a hydrostatic pressure profile (available for two-phase flow only)
- by specifying the constant pressure
- by specifying the neighboring cell pressure

Note

You can also specify a hydrostatic pressure profile at the pressure outlet for axisymmetric open channel flow. However, there are certain limitations as noted in *Limitations* (p. 1125).

Important

This option is not available in the case of three-phase flows since the pressure on the boundary is taken from the neighboring cell.

26.3.2.11. Limitations

The following list summarizes some issues and limitations associated with the open channel boundary condition.

- The conservation of the Bernoulli integral does not provide the conservation of mass flow rate for the pressure boundary. In the case of a coarser mesh, there can be a significant difference in mass flow rate from the actual mass flow rate. For finer meshes, the mass flow rate comes closer to the actual value. So, for problems having constant mass flow rate, the mass flow rate boundary condition is a better option. The pressure boundary should be selected when steady and nonoscillating drag is the main objective.
- Specifying the top boundary as the pressure outlet can sometimes lead to a divergent solution. This may be due to the corner singularity at the pressure boundary in the air region or due to the inability to specify local flow direction correctly if the air enters through the top locally.
- Only the heavier phase should be selected as the secondary phase.
- In the case of three-phase flows, only one secondary phase is allowed to enter through one inlet group (i.e., the mixed inflow of different secondary phases is not allowed).
- For axisymmetric flow, pressure and mass flow inlets do not support open channel boundary conditions.
- Open channel wave boundary conditions for velocity inlets do not support cases with axisymmetric flow.
- Axisymmetric open channel flow is available for two phase flow and the pressure outlet boundary.
- For axisymmetric open channel flow, the direction of gravity must be aligned with the direction of the axis.

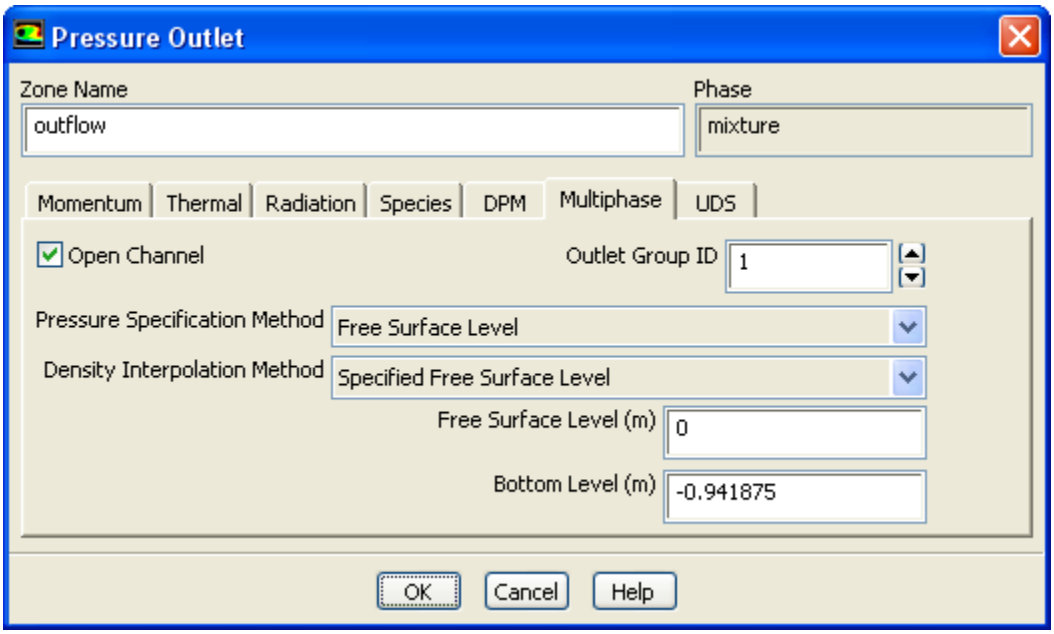
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.2.12. Choosing the Density Interpolation Method

For problems involving sub-critical flow, where the pressure specification method is the **Free Surface Level**, you can specify a **Density Interpolation Method**. Two options exist for the **Density Interpolation Method**:

- **Neighboring Cell Volume Fraction** is the default option, where the face density used in the hydrostatic profile is interpolated from the neighboring cell volume fraction.
- **Specified Free Surface Level** is the face density used in the hydrostatic profile and is interpolated from the specified free surface level.

Figure 26.16 Density Interpolation Method for the Free Surface Level



26.3.2.13. Recommendations for Setting Up an Open Channel Flow Problem

The following list represents a list of recommendations for solving problems using the open channel flow boundary condition:

- In the cases where the inlet group has a different inlet for each phase of fluid, then the parameter values (such as **Free Surface Level**, **Bottom Level**, and **Mass Flow Rate**) for each inlet should correspond to all other inlets that belong to the inlet group.
- The solution begins with an estimated pressure profile at the outlet boundary.

  In general, you can start the solution by assuming that the level of liquid at the outlet corresponds to the level of liquid at the inlet. The convergence and solution time is very dependent on the initial conditions. When the flow is completely subcritical (upstream and downstream), in marine applications for instance, the above approach is recommended.

  If the final conditions of the flow can be predicted by other means, the solution time can be significantly reduced by using the proper boundary condition.
- In the case of super-critical flow at the outlet, you can use the **From Neighboring Cell** option at the outlet, as it may provide better convergence.
- The initialization procedure is very critical in the open channel analysis.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are interested in the final steady state solution, then perform the following initialization procedure:

1. Initialize the domain by setting the volume fraction of the secondary phase to 0, and providing the inlet velocity.
2. Patch the domain using a volume fraction value of the secondary phase to 1, up to the **Free Surface Level** specified at the inlet.
3. Patch the inlet velocity again in the full domain.

If the **Free Surface Level** values are different at the inlet and outlet, then patching some regions with inlet **Free Surface Level** values and some regions with outlet **Free Surface Level** values could be useful for some problems.

The same steps for initialization are also recommended for transient flows, but now the initial conditions are dependent on the user.

- For the initial stability of the solution, a smaller time step is recommended. You can increase the time step once the solution becomes more stable.

26.3.3. Modeling Open Channel Wave Boundary Conditions

When modeling open channel wave boundary conditions, many of the variables that are used in open channel flow, also exist for open channel wave boundary conditions. You may have to refer to *Modeling Open Channel Flows* (p. 1121) for information about some of the settings.

To use the open channel wave boundary condition, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   General

2. Enable the **Volume of Fluid** model in the **Multiphase Model** dialog box.

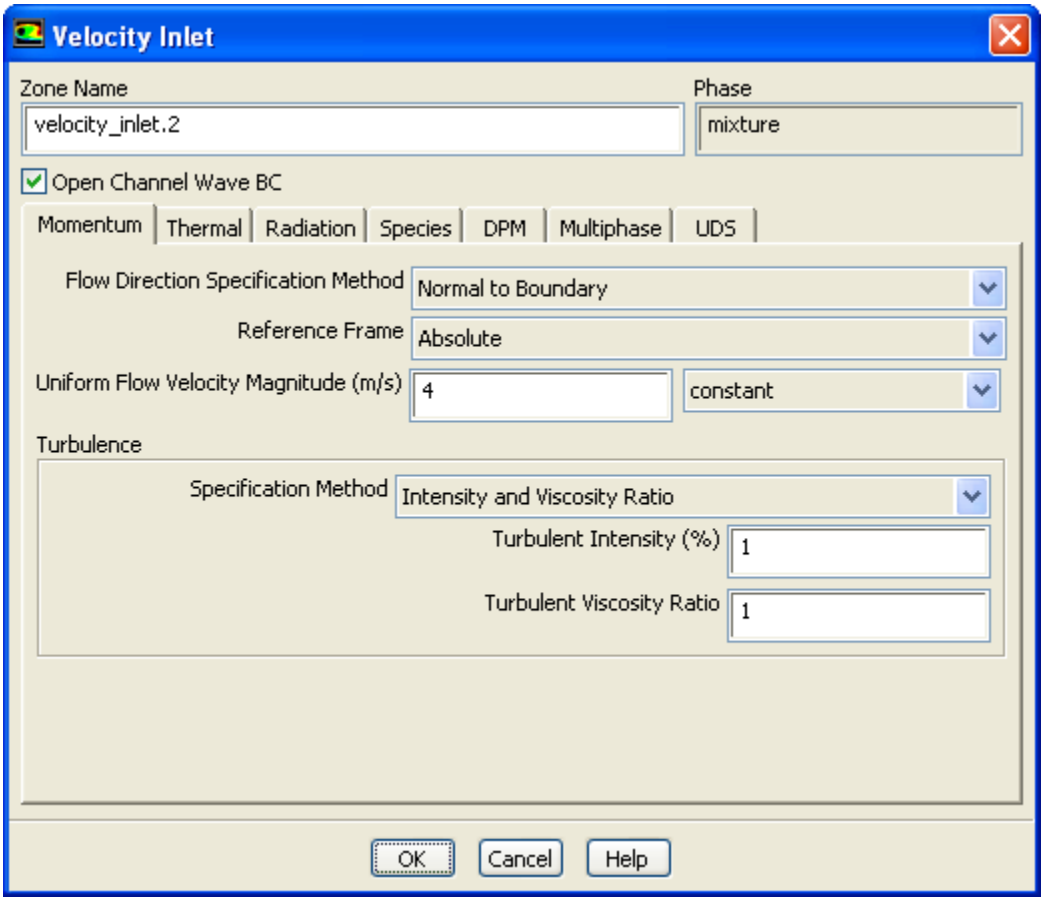
   Models → **Multiphase** → **Edit...**

3. Under **Scheme**, select either **Implicit** or **Explicit**.
4. Under **Volume Fraction Parameters**, select **Open Channel Wave BC**.

In order to set specific parameters for a particular boundary for open channel wave boundaries, enable the **Open Channel Wave BC** option in the **Velocity Inlet** boundary condition dialog box (*Figure 26.17* (p. 1128)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.17 The Velocity Inlet for Open Channel Wave BC



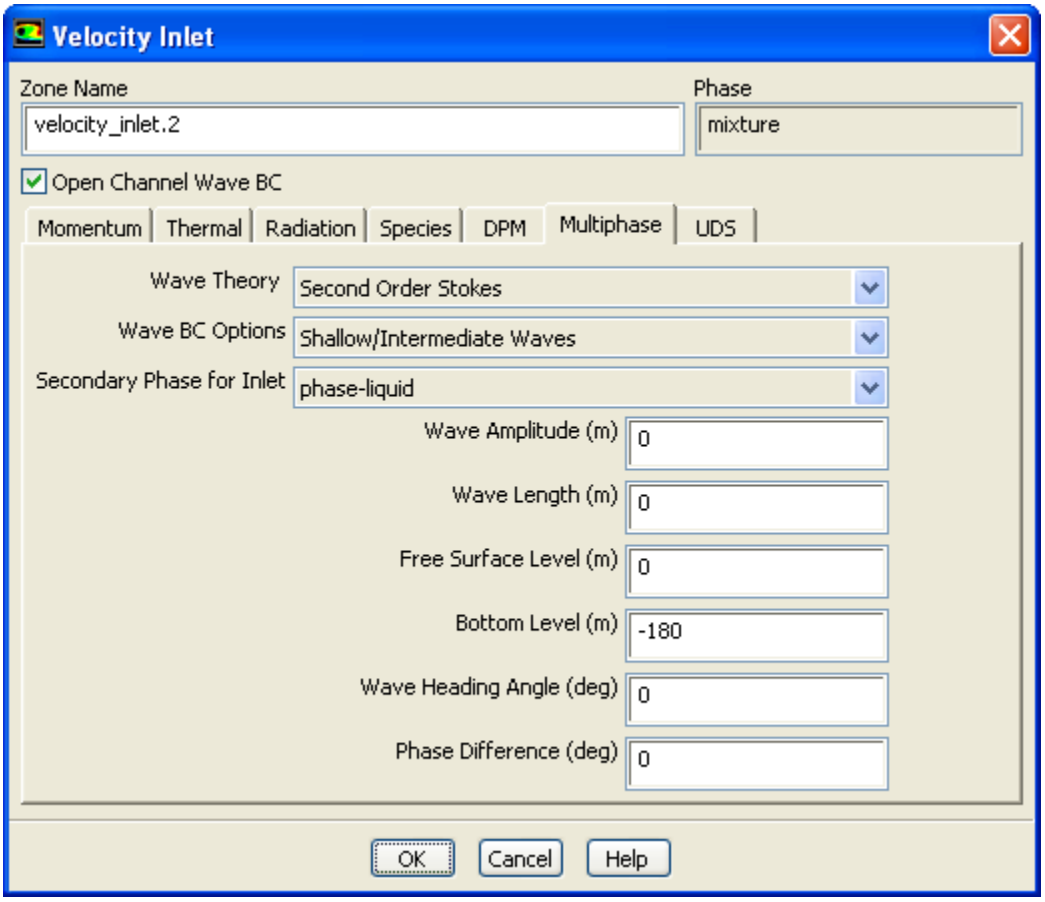
In the **Momentum** tab of the **Velocity Inlet** dialog box, enter the following wave specific variables:

- **Flow Direction Specification Method** which can be **Direction Vector** or **Normal to Boundary**.
- **Uniform Flow Velocity Magnitude** which can be specified as a **Constant** or a **New Input Parameter...**.

In the **Multiphase** tab (*Figure 26.18* (p. 1129)), you will specify the following:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.18 The Velocity Inlet for Open Channel Wave BC



- **Wave Theory** of which you have a choice of **First Order Airy** (the default), **Second Order Stokes**, **Third Order Stokes**,**Fourth Order Stokes**, and **Fifth Order Stokes**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.
- **Wave BC Options** of which you have a choice of **Shallow/Intermediate Waves** or **Short Gravity Waves**. Information about the two types of waves is available in Open Channel Wave Boundary Conditions in the Theory Guide. Note that the short gravity waves expression is derived under the assumption of infinite liquid height.
- **Secondary Phase for Inlet** is where the specified parameters are valid only for one secondary phase. In case of a three-phase flow, select the corresponding secondary phase from this list.
- **Wave Amplitude** is the amplitude of the shallow or short gravity waves.
- **Wave Length** is the wavelength of the shallow or short gravity waves.
- **Free Surface Level** is the same definition as for open channel flow, see *Modeling Open Channel Flows* (p. 1121).
- **Bottom Level** is the same definition as for open channel flow, see *Modeling Open Channel Flows* (p. 1121), and is valid only for shallow waves. The bottom level is used for calculating the liquid height.
- **Wave Heading Angle** is the angle between the wave front and the flow direction, in the plane of the flow direction and cross-flow direction.
- **Phase Difference** is the phase angle by which one periodic disturbance or wave front lags behind or precedes another in time or space.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A useful text command used to print out a summary of the open channel wave boundary condition settings is `define/boundary-conditions/open-channel-wave-settings`. Below is a sample of the output displayed in the text user interface:

```
/define/boundary-conditions> open-channel-wave-settings


Wave Input Analysis for Velocity Inlet : Thread ID = 4
*******************************************************
Current Settings :
------------------
Second Order Stokes, Shallow/Intermediate Waves
Wave Amplitude (A) = 7.3711, Wave Length (L) = 189.2780, Depth (h) = 180.0000
Wave Height (H) = 14.7422, Wave Steepness (H/L) = 0.0779
Ursell Number (Ur = H*L*L/(h*h*h)) = 0.0906

Checking parameters within wave breaking limit :
------------------------------------------------
Wave Height/Depth ratio
H/h = 0.0819, Max limit = 0.78
H/h ratio within wave breaking limit

Wave_Height/Wave_Length ratio
H/L = 0.0779, Max limit = 0.1420
H/L ratio within wave breaking limit

Checking Stability Criterion based on Ursell number
---------------------------------------------------
Ursell number = 0.0906, Max Limit = 26.0000
Ursell number within stability limit

Summary :
---------
Checks successful

Recommendations :
------------------
Preferred Wave theories : 3rd to 5th Order Stokes
```

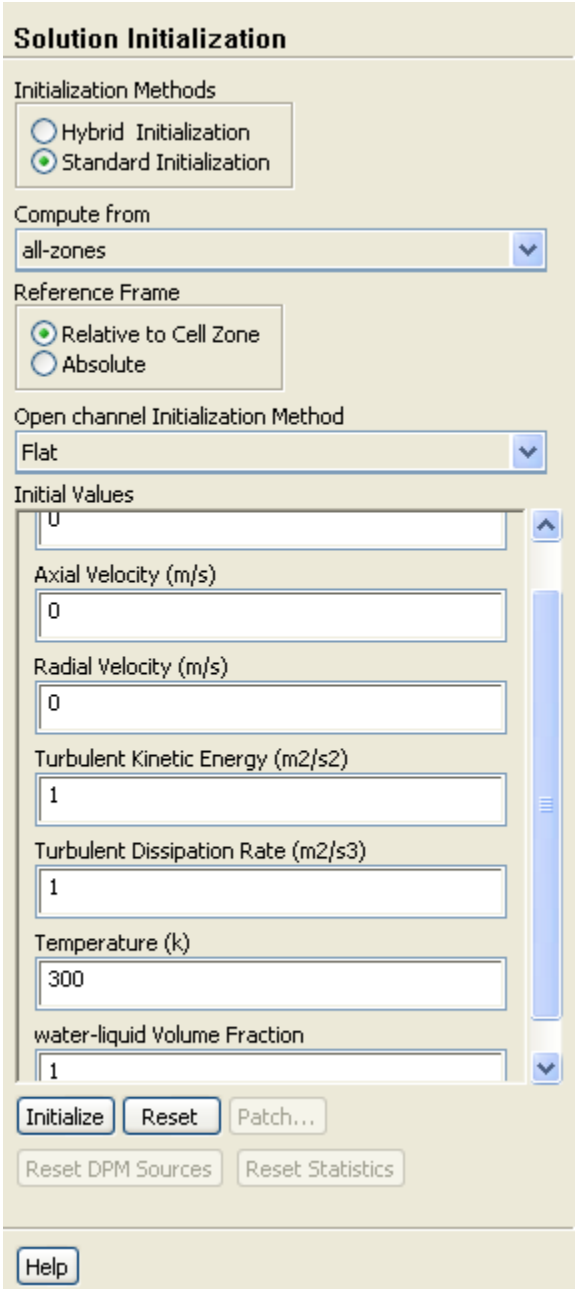
26.3.4. Recommendations for Open Channel Initialization

Once you have selected either the **Open Channel Flow** or the **Open Channel Wave BC** option in the **Multiphase Model** dialog box, then the **Open Channel Initialization Method** drop-down list appears in the **Solution Initialization** task page.

Solution Initialization

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.19 The Solution Initialization Task Page



Select an inlet zone from the **Compute from** drop-down list. You can now make your selection from the **Open Channel Initialization Method** drop-down list. If only the **Open Channel Flow** option was enabled, then you only have a choice of **None** or **Flat**. If you enabled **Open Channel Wave BC**, then your choices are **None**, **Flat**, or **Wavy**. The default initialization method is **None**.

If you initialize the solution using **None**, it has no effect as it does not use any open channel information from the selected zone. The **Open Channel Initialization Method** comes into effect when you select either **Flat** or **Wavy**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

This initialization is only valid for pressure-inlets, pressure outlets, and mass-flow inlets for open channel flow and velocity inlets for open channel wave boundary conditions. If the selected inlet zone does not have either open channel flow or open channel wave boundary conditions, ANSYS FLUENT will report an error message after you initialize the flow with open channel initialization method of **Flat** or **Wavy**.

For open channel initialization from the pressure outlet boundary, the hydrostatic pressure profile based on the **Free Surface Level** is patched in the domain. The volume fraction in the domain is patched based on **Free Surface Level** provided at the pressure outlet boundary. To patch velocity and other variables, the values in the **Solution Initialization** task page will be used.

Important

- Open channel initialization from the pressure outlet boundary is only supported for two phase flow.
- The pressure specification methods, with the exception of **Free Surface Level**, are not supported for open channel initialization from the pressure outlet boundary.

Initialization will result in the volume fraction, X, Y, and Z velocities, and pressure being patched in the domain. The volume fraction will be patched in the domain based on the free surface level of the selected zone from the **Compute from** list. The velocities in the domain will be patched assuming the constant value provided for the velocity magnitude in the selected zone.

Important

If you specify a profile for the velocity magnitude or direction vectors, the initialization will select the value for the velocity magnitude and direction vectors from only one face. Therefore the initialization may be inaccurate. However, generally, open channel inputs for velocity magnitude and direction vectors are constant.

The pressure which is patched is the hydrostatic pressure based on the free surface level specified in the selected zone.

You can use the following text command for open channel automatic initialization:

`solve` → `initialize` → `open-channel-auto-init`

When prompted, set up the following parameters:

boundary thread id
Enter the thread id for the boundary to be selected for open channel automatic initialization.

flat free surface initialization
This option is available for both open channel flow and open channel wave boundary conditions.

wavy free surface initialization
This option appears only for open channel wave boundary conditions, when flat free surface initialization is not selected.

The steps to be followed for open channel automatic initialization are

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Compute defaults based on valid open channel boundary thread. This step is required for better initialization of turbulence parameters based on uniform velocity magnitude.

   `solve` → `initialize` → `compute defaults`

2. This would provide information about the selected boundary and type of initialization.

   `solve` → `initialize` → `open-channel-auto-init`

3. Initialize

   `solve` → `initialize` → `initialize`

26.3.5. Numerical Beach Treatment for Open Channels

In certain applications, it is desirable to suppress numerical reflection near the outlet boundary for wave dampening. To understand the theory involved in this application, please refer to Numerical Beach Treatment

To include numerical beach in your simulation, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   General

2. Enable the **Volume of Fluid** model in the **Multiphase Model** dialog box.

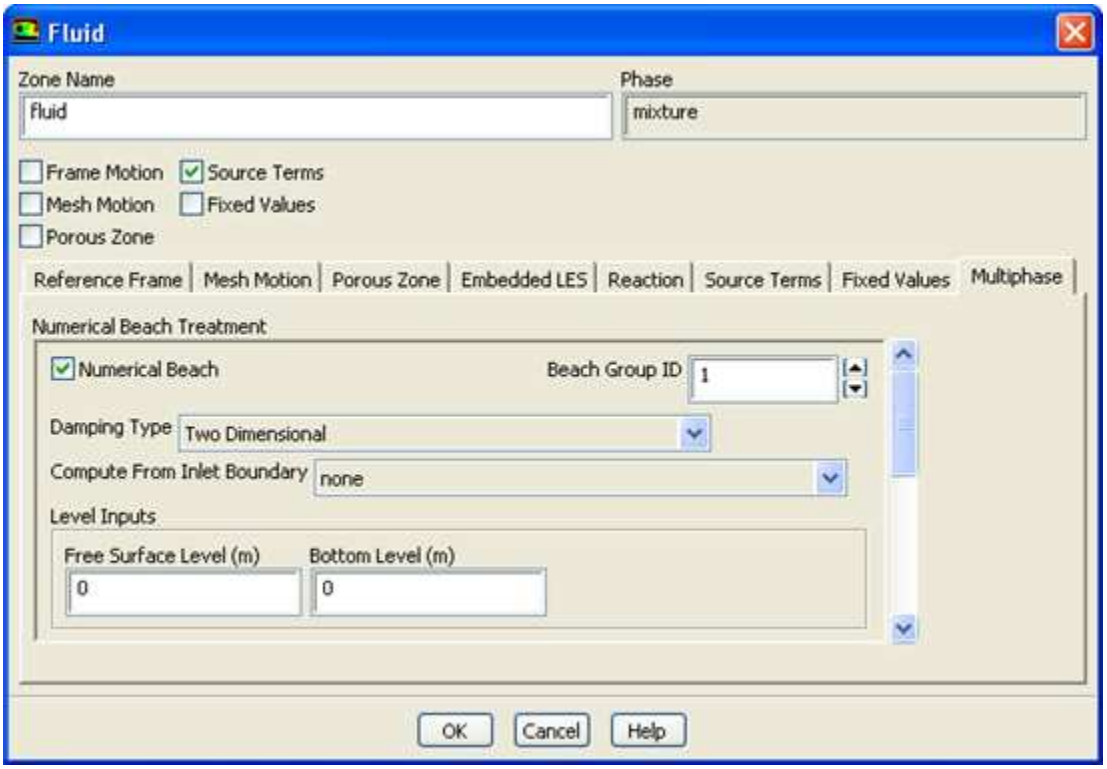
   Models → **Multiphase** → **Edit...**

3. Under **Scheme**, select either **Implicit** or **Explicit**.
4. Select **Open Channel Flow** and/or **Open Channel Wave BC**.

In order to set the numerical beach parameters for a fluid zone, go to the **Fluid** dialog box (*Figure 26.20* (p. 1134)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.20 The Fluid Dialog Box Displaying Numerical Beach



In the **Multiphase** tab of the **Fluid** dialog box, enable the **Numerical Beach** option and enter the following:

- **Beach Group ID** represents the cell zones sharing the damping length containing the same input parameters.
- **Damping Type** allows you to choose between **Two Dimensional** and **One Dimensional**.
  - **Two Dimensional** is the damping treatment in the flow and gravity direction.
  - **One dimensional** is the damping treatment in the flow direction.
- **Compute from Inlet Boundary** is set to **none** by default. If there are available open channel boundaries (velocity-inlet, pressure-inlet, and mass-flow-inlet), boundary names are added to the drop-down list. If you select a boundary from the list, the **Level Inputs** and the **Damping Length Inputs in Flow Direction** will be updated in the interface. You have the option to overwrite the updated inputs with values that are more applicable to your simulation.
- **Level Inputs** is only available for the **Two Dimensional** damping type.
  - **Free Surface Level** is the same definition as for open channel flow, see *Modeling Open Channel Flows* (p. 1121).
  - **Bottom Level** is the same definition as for open channel flow, see *Modeling Open Channel Flows* (p. 1121). During the automatic calculation from the velocity inlet boundary for short gravity waves, this parameter is updated under some assumptions, as noted in *Solution Strategies* (p. 1135). The bottom level is used for calculating the liquid height.
- **Flow Direction** is the X, Y, and Z (for 3D) components.
- **Damping Length Inputs in Flow Direction** are required to calculate the start and end points of the damping length in the flow direction.
  - **Damping Length Specification** is only available if **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box. There are two options you can choose from:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **End Point and Wave Lengths** is the default option.

→ **End and Start Points** are the limits of the damping zone.

– **End Point** is the end point of the damping zone. **End Point** is updated automatically if the boundary is selected from the **Compute from Inlet Boundary** drop-down list.

Note

The calculated value is assumed based on the domain extents for all the cell zones, and you should enter your own value if needed.

The end point is calculated by taking the dot product of the flow direction $x_e = \vec{X}_e . \hat{x}$

Here, x_e is the end point and $\vec{X}_e$ is the position vector in the plane perpendicular to the flow direction $\hat{x}$.

– **Start Point** is the starting point in the flow direction.

The start point is calculated by taking the dot product of the flow direction $x_s = \vec{X}_s . \hat{x}$

Here, x_s is the end point and $\vec{X}_e$ is the position vector in the plane perpendicular to the flow direction $\hat{x}$.

– **Wave Length** is updated automatically if the boundary is selected from the **Compute from Inlet Boundary** drop-down list.

– **Number Of Wave Lengths** is set to 2 by default for the calculation of the damping length.

- **Damping Resistance** is the resistance per unit length.

26.3.5.1. Solution Strategies

Below are some helpful key points when using the **Numerical Beach** option:

1. The **Compute from Inlet Boundary** drop-down list is a convenient feature as it provides automatic inputs, which you should check to make sure the values are reasonable. Some of the provided values are calculated under certain assumptions:
   a. The bottom level, in the case of short gravity waves, is selected in the velocity inlet dialog box for open channel wave boundary conditions. Since there is no user input for this parameter, ANSYS FLUENT calculates the bottom level as the value of the free surface level less 2.5 times the wave length. The assumption is that the liquid height is 2.5 times the wave length.
   b. The end point domain extents in the flow direction and includes all the cell zones.
2. For manual inputs of start and end points, you should verify that these points are calculated correctly in the flow direction. To check for their validity, subtracting the start point from the end point should give you a positive value.
3. For manual inputs of **Free Surface Level** and **Bottom Level**, you should verify that these points are calculated correctly in the direction opposite to gravity. To check for their validity, subtracting the bottom level from the free surface level should give you a positive value.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In case you create separate damping zones, but the damping length is not sufficient to suppress the waves, clubbing of the beach could be done by selecting the same beach ID for other cell zones. In this case, both the cell zones would share the same information.
5. In case of separate damping zones (without clubbing), you should verify that the damping length is more than or equal to the specified number of wave lengths for the calculation of the start point.
6. Damping resistance should be chosen accordingly: too much or too little damping could affect the wave profiles in a no-damping zone. For higher order waves, a default value of 10 was satisfactory, but this value is case specific and should be tuned to get better results with minimal reflection.
7. Steep damping at the beginning of the damping zone could affect the wave profile just before the damping zone.
8. It is recommended to use a coarse mesh in the damping zone with increased coarseness towards the end of the damping zone.

26.3.6. Defining the Phases for the VOF Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction in a VOF calculation are provided below.

Important

In general, you can specify the primary and secondary phases whichever way you prefer. It is a good idea, especially in more complicated problems, to consider how your choice will affect the ease of problem setup. For example, if you are planning to patch an initial volume fraction of 1 for one phase in a portion of the domain, it may be more convenient to make that phase a secondary phase. Also, if one of the phases is a compressible ideal gas, it is recommended that you specify it as the primary phase to improve solution stability.

Important

Recall that only one of the phases can be a compressible ideal gas. Be sure that you do not select a compressible ideal gas material (i.e., a material that uses the compressible ideal gas law for density) for more than one of the phases. See *Modeling Compressible Flows* (p. 1144) and *Modeling Compressible Flows* (p. 1153) for details.

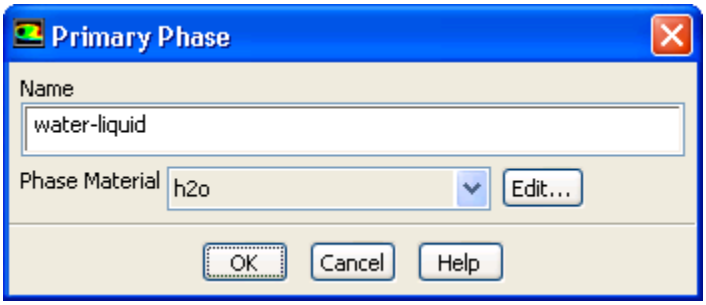
26.3.6.1. Defining the Primary Phase

To define the primary phase in a VOF calculation, perform the following steps:

1. Select **phase-1** in the **Phases** list.
2. Click **Edit...** to open the **Primary Phase** dialog box (*Figure 26.21* (p. 1137)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.21 The Primary Phase Dialog Box



3. In the **Primary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**.
   a. Click **Edit...**, and the **Edit Material** dialog box will open.
   b. In the **Edit Material** dialog box, check the properties, and modify them if necessary. (See *Physical Properties* (p. 385) for general information about setting material properties, *Modeling Compressible Flows* (p. 1144) for specific information related to compressible VOF calculations, and *Modeling Solidification/Melting* (p. 1145) for specific information related to melting/solidification VOF calculations.)

   Important

   If you make changes to the properties, remember to click **Change** before closing the **Edit Material** dialog box.

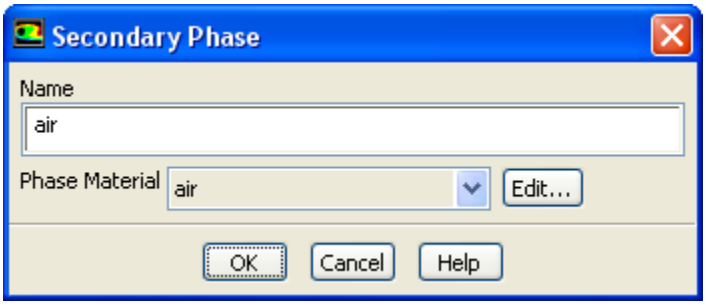
6. Click **OK** in the **Primary Phase** dialog box.

26.3.6.2. Defining a Secondary Phase

To define a secondary phase in a VOF calculation, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the **Secondary Phase** dialog box (*Figure 26.22* (p. 1137)).

Figure 26.22 The Secondary Phase Dialog Box for the VOF Model



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Define the material properties for the **Phase Material**, following the procedure outlined above for setting the material properties for the primary phase.
6. Click **OK** in the **Secondary Phase** dialog box.

26.3.6.3. Including Surface Tension and Adhesion Effects

As discussed in When Surface Tension Effects Are Important in the Theory Guide, the importance of surface tension effects depends on the value of the capillary number, Ca (defined by Equation 17–24 in the Theory Guide), or the Weber number, We (defined by Equation 17–25 in the Theory Guide). Surface tension effects can be neglected if Ca $\gg 1$ or We $\gg 1$.

Several surface tension options are provided through the text user interface (TUI) using the `solve/set/surface-tension` command :

`solve` $\rightarrow$ `set` $\rightarrow$ `surface-tension`

The `surface-tension` command prompts you for the following information:

- whether you require node-based smoothing

  The default value is `yes`, indicating that node-based smoothing will be used. Note that if you are reading in a case that was created in versions prior to ANSYS FLUENT 13, then cell-based smoothing will be used by default for the VOF calculations.
- the number of smoothings

  The default value is `1`. A higher value can be used in case of tetrahedral and triangular meshes in order to reduce any spurious velocities.
- the smoothing relaxation factor

  The default is `1`. This is useful in the cases where VOF smoothing causes a problem (e.g., liquid enters through the inlet with wall adhesion on).
- whether you want to use VOF gradients at the nodes for curvature calculations

  With this option, ANSYS FLUENT uses VOF gradients directly from the nodes to calculate the curvature for surface tension forces. The default is `yes` which produces better results with surface tension compared to gradients that are calculated at the cell centers.

Important

Note that the calculation of surface tension effects will be more accurate if you use a quadrilateral or hexahedral mesh in the area(s) of the computational domain where surface tension is significant. If you cannot use a quadrilateral or hexahedral mesh for the entire domain, then you should use a hybrid mesh, with quadrilaterals or hexahedra in the affected areas.

ANSYS FLUENT uses node based smoothing for smoothed volume fraction and node based gradients for smoothed curvature calculation to provide better accuracy and robustness.

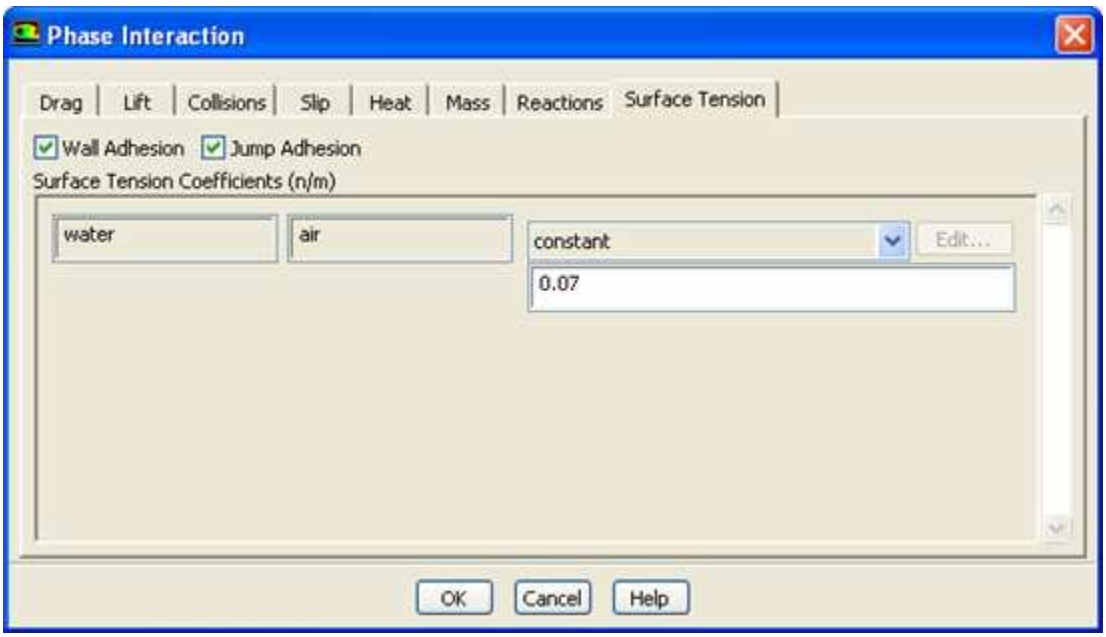
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Pressure jump caused by the surface tension is discontinuous in nature, and also it acts locally at the interface. These numerical difficulties are overcome in an approximate manner by considering the smoothed distribution of the volume fraction field within the finite interfacial width. Smoothing procedures, in general, are mesh dependent. Therefore, the amount and nature of the smoothing procedure could have a significant effect on the results for surface tension cases.

If you want to include the effects of surface tension along the interface between one or more pairs of phases, as described in Surface Tension and Adhesion in the Theory Guide, click **Interaction...** to open the *Phase Interaction Dialog Box* (p. 1791) (*Figure 26.23* (p. 1139)).

Figure 26.23 The Phase Interaction Dialog Box for the VOF Model (Surface Tension Tab)



Perform the following steps to include surface tension (and, if appropriate, wall adhesion) effects along the interface between one or more pairs of phases:

1. Click the **Surface Tension** tab.
2. For each pair of phases between which you want to include the effects of surface tension, specify a constant surface tension coefficient. Alternatively you can specify a temperature dependent, polynomial, piece-wise polynomial, piecewise linear, or a user-defined surface tension coefficient. See Surface Tension and Adhesion in the Theory Guide for more information on surface tension, and the UDF Manual for more information on user-defined functions. All surface tension coefficients are equal to 0 by default, representing no surface tension effects along the interface between the two phases.

Important

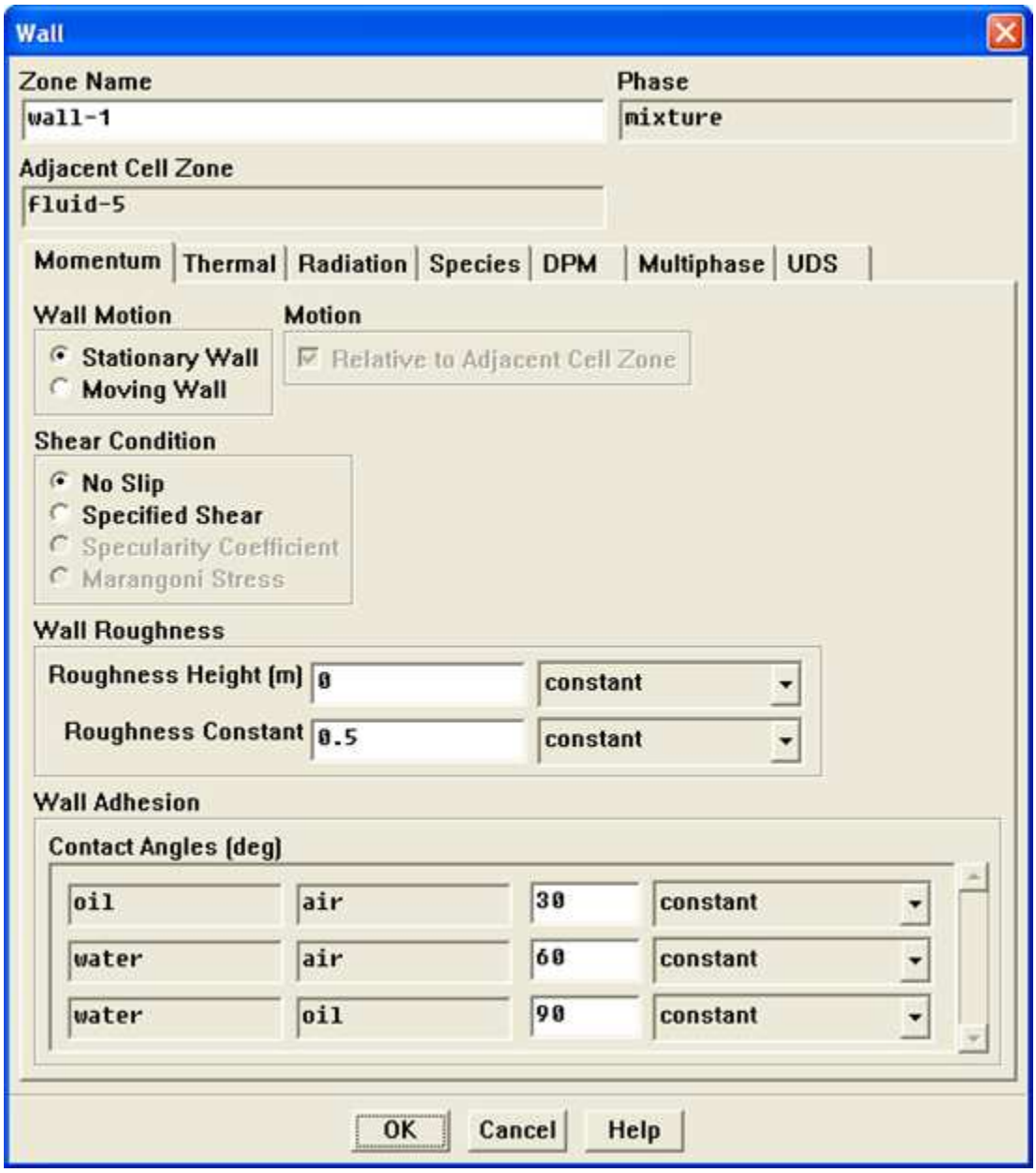
For calculations involving surface tension, it is recommended that you also turn on the **Implicit Body Force** treatment for the **Body Force Formulation** in the **Multiphase Model** dialog box. This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and surface tension forces in the momentum equations. See *Including Body Forces* (p. 1099) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If you want to include wall adhesion, enable the **Wall Adhesion** option. When **Wall Adhesion** is enabled, you will need to specify the contact angle at each wall as a boundary condition (as described in *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107)).

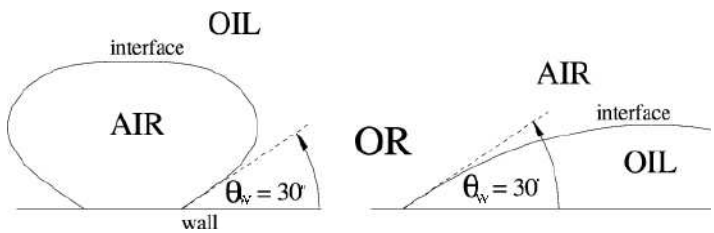
   The contact angle θ_w is the angle between the wall and the tangent to the interface at the wall, measured inside the *phase listed in the left column* under **Wall Adhesion** in the **Momentum** tab of the **Wall** dialog box. For example, if you are setting the contact angle between the oil and air phases in the **Wall** dialog box shown in *Figure 26.24* (p. 1140), θ_w is measured inside the oil phase, as seen in *Figure 26.25* (p. 1141). For more information, refer to Wall Adhesion in the Theory Guide.

Figure 26.24 The Wall Dialog Box for a Mixture in a VOF Calculation with Wall Adhesion



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.25 Measuring the Contact Angle



4. If you want to include jump adhesion, enable the **Jump Adhesion** option. When **Jump Adhesion** is enabled, you will need to specify the contact angle at the porous jump in the **Porous Jump** dialog box (as described in *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107)). The contact angle specification and measurement for jump adhesion is similar to that of wall adhesion (as explained in step 3 above).

Important

Jump adhesion only supports the cell based smoothing option for surface tension. When the **Jump Adhesion** option is enabled in the **Phase Interaction** dialog box, the correct settings are automatically set for surface tension. To view the surface tension settings, use the `solve/set/surface-tension` text command.

26.3.6.4. Discretizing Using the Phase Localized Compressive Scheme

Most of the interface capturing or tracking schemes, which are common in literature, can simulate either diffused interface modeling *or* sharp interface modeling. There are a number of applications where you may be interested in modeling diffused *and* sharp interfaces in different regimes, which results in the need for a unique discretization procedure to handle such applications.

Below are some examples of such applications:

- **Air bubble rise in water-solid slurry:** Diffused interface modeling is required for the water and solid, and sharp interface modeling for the air bubble is desirable. A sharpening scheme would cause undesirable effects for the modeling of the diffused interface between water and solids and a diffusive scheme would not be able to maintain the sharp interface for bubble rising in the slurry.
- **Evaporation/condensation in a tank partially filled with water and being heated at the bottom:** Diffused interface modeling is required for the liquid-vapor and sharp interface modeling for the water-air regime is desirable. A sharpening scheme, would not be able to simulate the diffused phenomena of evaporation/condensation for the liquid- vapor phase and a diffusive scheme would not be able to maintain the sharp interface between water and air.
- **Air jet penetrating through layer of liquids:** You may be interested in diffused jet modeling and sharp interface modeling between the liquid layers. A sharpening scheme would not be able to maintain the continuous air stream and a diffusive scheme would not be able to maintain the sharp interfaces between the layers of liquids.

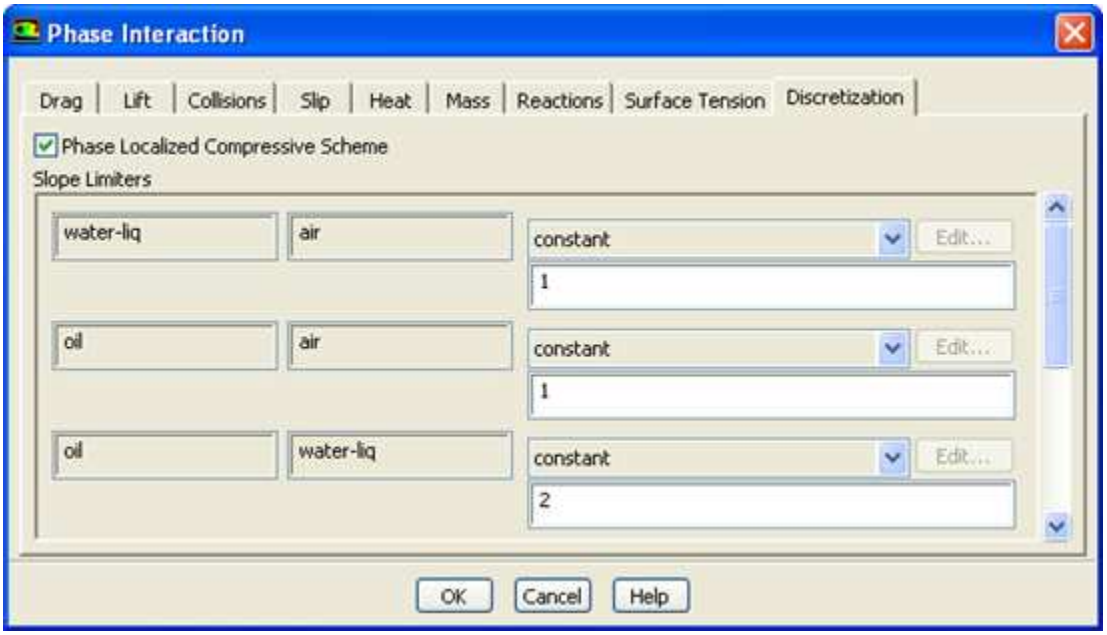
Using the Modified HRIC and Compressive schemes for such applications would result in the HRIC scheme producing undesirable sharpening of the dispersed phases and undesirable diffusion of the continuous phases, whereas the Compressive scheme would produce undesirable sharpening of the dispersed phases.

Therefore, the phase localized compressive scheme is particularly useful in cases where the desirable behavior is such that you have diffused modeling of dispersed phases and sharp modeling of continuous phases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In ANSYS FLUENT, diffusive and anti-diffusive discretization procedures can be used across the distinct interfaces, which share a pair of phases. This functionality is provided through the compressive discretization scheme, where the degree of diffusion or sharpness is controlled through the value of the slope limiters. The theory used is described in The Compressive and Zonal Discretization Schemes.

Figure 26.26 The Phase Interaction Dialog Box for the VOF Model (Discretization Tab)



To use the phase localized compressive scheme, perform the following steps:

1. Click the **Discretization** tab.

   Note

   The **Discretization** tab is only available if the VOF model is enabled, or if the Eulerian multiphase model is selected with the immiscible fluid model enabled.

2. Enable **Phase Localized Compressive Scheme**.

   Important

   When this option is enabled, the **Compressive** spatial discretization scheme is automatically selected for the **Volume Fraction** in the **Solution Methods** dialog box.

3. For each pair of phases, specify the value of the slope limiter. You can enter a value of 0, 1, or 2, or any value between 0 and 2. Please refer to *Table 26.8: Slope Limiter Discretization Scheme* (p. 1142) to equate each value of the slope limiter with a discretization scheme.

Table 26.8 Slope Limiter Discretization Scheme

Slope Limiter Value $\beta_{\max}$	Scheme
0	first order upwind

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Slope Limiter Value $\beta_{\max}$	**Scheme**
1	second order reconstruction bounded by the global minimum/maximum of the volume fraction
2	compressive
$0<\beta<1$ and $1<\beta<2$	blended: where a value between 0 and 1 means blending of the first order and second order and a value between 1 and 2 means blending of the second order and compressive scheme

When using the **Phase Localized Compressive Scheme**, please keep in mind the following:

- The minimum limit for the slope limiter is 0, whereas the maximum is 2.
- The slope limiter can be interpreted as the degree of compression/anti-diffusion, where 0 demonstrates the minimum compression and 2 demonstrates the maximum compression.
- For interfaces sharing the diffused phases, a slope limiter value of 2 should not be used. A value between 0 and 1 is recommended.
- For interfaces sharing the continuous phases, any value between 0 and 2 can be used depending on the application.
- For interfaces sharing the continuous and diffused phases, any value between 0 and 2 can be used based on the application.
- If you want variable discretization behavior for an interface between two phases, you can do so via the `DEFINE_PROPERTY` UDF.

Note

– If the interface has a transition from sharp to diffused modeling, you should not experience any problems.

– If the interface has a transition from diffused to sharp modeling, this transition should be smooth by gradually varying the value of the slope limiter within some transition zone.

26.3.7. Setting Time-Dependent Parameters for the VOF Model

If you are using the time-dependent volume fraction formulation in ANSYS FLUENT, an explicit solution for the volume fraction is obtained either once each time step or once each iteration, depending upon your inputs to the model. You also have control over the time step used for the volume fraction calculation.

To compute a time-dependent VOF solution, you will need to enable the **Transient** option in the **General** task page (and choose the appropriate **Transient Formulation** in the **Solution Methods** task page, as discussed in *User Inputs for Time-Dependent Problems* (p. 1256)).

There are two inputs for the time-dependent calculation for the VOF model:

- By default, ANSYS FLUENT will solve the volume fraction equation(s) once for each time step. This means that the convective flux coefficients appearing in the other transport equations will not be completely updated each iteration, since the volume fraction fields will not change from iteration to iteration.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want ANSYS FLUENT to solve the volume fraction equation(s) at every iteration within a time step, first make sure that you have selected **Volume of Fluid** from the **Model** list in the **Multiphase Model** dialog box, and then enter the following text command in the console:

`define` → `models` → `multiphase` → `volume-fraction-parameters`

When prompted to `solvevofevery iteration?`, enter `yes`.

When ANSYS FLUENT solves these equations every iteration, the convective flux coefficients in the other transport equations will be updated based on the updated volume fractions at each iteration. This choice is the less stable of the two, and requires more computational effort per time step than the default choice.

Important

If you are using sliding meshes, or dynamic meshes with layering and/or remeshing, using the `solve vof every iteration?` option will yield more accurate results, although at a greater computational cost.

- When ANSYS FLUENT performs a time-dependent VOF calculation, the time step used for the volume fraction calculation will not be the same as the time step used for the rest of the transport equations. ANSYS FLUENT will refine the time step for VOF automatically, based on your input for the maximum **Courant Number** allowed near the free surface. The Courant number is a dimensionless number that compares the time step in a calculation to the characteristic time of transit of a fluid element across a control volume:

$$\frac{\Delta t}{\Delta x_{cell}/v_{fluid}} \tag{26–4}$$

In the region near the fluid interface, ANSYS FLUENT divides the volume of each cell by the sum of the outgoing fluxes. The resulting time represents the time it would take for the fluid to empty out of the cell. The smallest such time is used as the characteristic time of transit for a fluid element across a control volume, as described above. Based upon this time and your input for the maximum allowed **Courant Number** in the *Multiphase Model Dialog Box* (p. 1642), a time step is computed for use in the VOF calculation. For example, if the maximum allowed Courant number is 0.25 (the default), the time step will be chosen to be at most one-fourth the minimum transit time for any cell near the interface.

Note that these inputs are not required when the implicit scheme is used.

26.3.8. Modeling Compressible Flows

If you are using the VOF model for a compressible flow, note the following:

- Only one of the phases can be defined as a compressible ideal gas (i.e., you can select the ideal gas law for the density of only one phase's material). There is no limitation on using compressible liquids using user-defined functions.
- When using the VOF model, for stability reasons, it is better (although not required) if the primary phase is a compressible ideal gas.
- If you specify the total pressure at a boundary (e.g., for a pressure inlet or intake fan) the specified value for temperature at that boundary will be used as total temperature for the compressible phase, and as static temperature for the other phases (which are incompressible).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For each mass flow inlet, you will need to specify mass flow or mass flux for each individual phase.

Important

Note that if you read a case file that was set up in a version of ANSYS FLUENT previous to 6.1, you will need to redefine the conditions at the mass flow inlets. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for more information on defining conditions for a mass flow inlet in VOF multiphase calculations.

See *Compressible Flows* (p. 503) for more information about compressible flows.

26.3.9. Modeling Solidification/Melting

If you are including melting or solidification in your VOF calculation, note the following:

- It is possible to model melting or solidification in a single phase or in multiple phases.
- For phases that are *not* melting or solidifying, you must set the latent heat (L), liquidus temperature ($T_{liquidus}$), and solidus temperature ($T_{solidus}$) to zero.

See *Modeling Solidification and Melting* (p. 1201) for more information about melting and solidification.

26.4. Setting Up the Mixture Model

For background information about the mixture model and the limitations that apply, refer to Overview in the Theory Guide.

For additional information, please see the following sections:

26.4.1. Defining the Phases for the Mixture Model
26.4.2. Including Cavitation Effects
26.4.3. Modeling Compressible Flows

26.4.1. Defining the Phases for the Mixture Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction for a mixture model calculation are provided below.

Important

Recall that only one of the phases can be a compressible ideal gas. Be sure that you do not select a compressible ideal gas material (i.e., a material that uses the compressible ideal gas law for density) for more than one of the phases. See *Modeling Compressible Flows* (p. 1153) for details.

26.4.1.1. Defining the Primary Phase

The procedure for defining the primary phase in a mixture model calculation is the same as for a VOF calculation. See *Defining the Primary Phase* (p. 1136) for details.

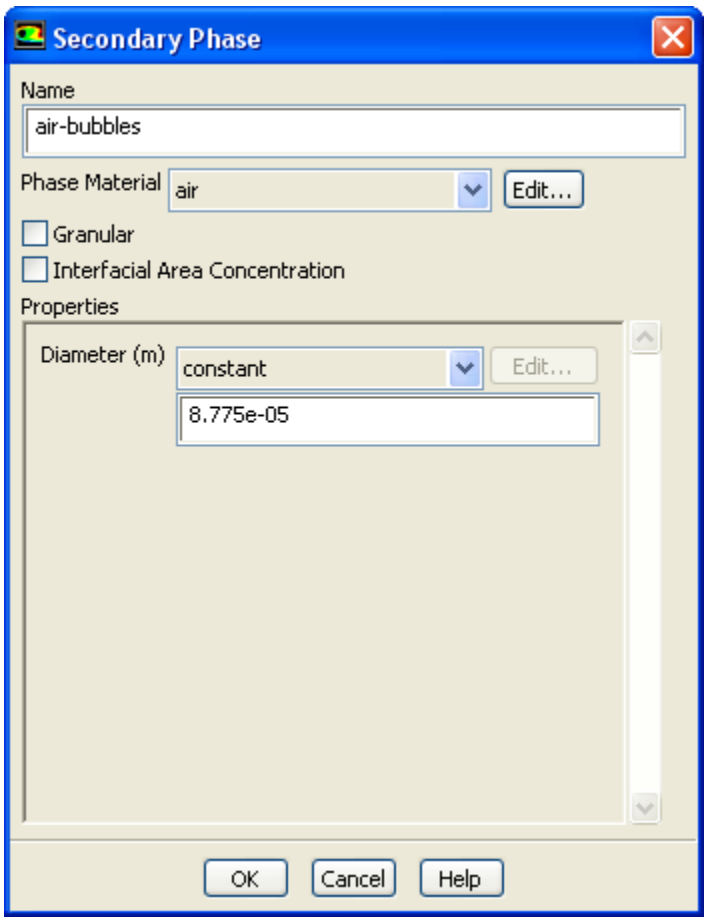
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.4.1.2. Defining a Nongranular Secondary Phase

To define a nongranular (i.e., liquid or vapor) secondary phase in a mixture multiphase calculation, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the *Secondary Phase Dialog Box* (p. 1785) (*Figure 26.27* (p. 1146)).

Figure 26.27 The Secondary Phase Dialog Box for the Mixture Model



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see *Defining the Primary Phase* (p. 1136)). For a particulate phase (which must be placed in the fluid materials category, as mentioned in *Steps for Using a Multiphase Model* (p. 1091)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.
6. In the **Secondary Phase** dialog box, specify the **Diameter** of the bubbles, droplets, or particles of this phase (d_p in Equation 17–88 in the Theory Guide). You can specify a constant value, or use a user-defined function. See the UDF Manual for details about user-defined functions. Note that when you are using the mixture model without slip velocity, this input is not necessary, and it will not be available to you.
7. Click **OK** in the **Secondary Phase** dialog box.

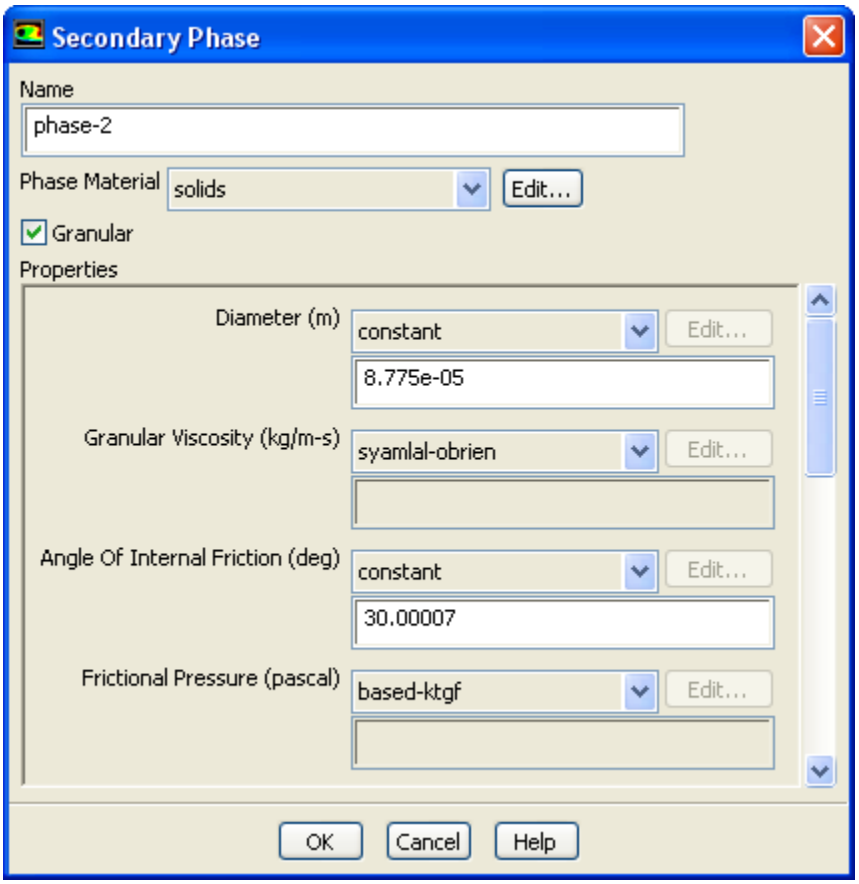
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.4.1.3. Defining a Granular Secondary Phase

To define a granular (i.e., particulate) secondary phase in a mixture model multiphase calculation, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the *Secondary Phase Dialog Box* (p. 1785) (*Figure 26.28* (p. 1147)).

Figure 26.28 The Secondary Phase Dialog Box for a Granular Phase Using the Mixture Model



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see *Defining the Primary Phase* (p. 1136)). For a granular phase (which must be placed in the fluid materials category, as mentioned in *Steps for Using a Multiphase Model* (p. 1091)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

Important

Note that all properties for granular flows can utilize user-defined functions (UDFs).

See the UDF Manual for details about user-defined functions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Enable the **Granular** option.
7. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

Diameter
specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the UDF Manual for details about user-defined functions.

Granular Viscosity
specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 17–96 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 17–98 in the Theory Guide, select **gidaspow** to compute the value using Equation 17–99 in the Theory Guide, or select **user-defined** to use a user-defined function.

Frictional Pressure
specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17–208 in the Theory Guide, **syamlal-obrien** to apply Equation 17–148 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [*20*] (p. 2210). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the UDF manual for information on hooking a UDF.

Frictional Modulus
is defined as

$$G = \frac{\partial P_{friction}}{\partial \alpha_{friction}} \qquad \textbf{(26–5)}$$

with $G \geq 0$, which is the **derived** option. You can also specify a **user-defined** function for the frictional modulus.

Friction Packing Limit
specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5. This is only a general rule of thumb as there may be other factors involved.

Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. Choose either the **algebraic**, the **constant**, or **user-defined** option.

Solids Pressure
specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution
specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastoopour**, or a **user-defined** option.

Elasticity Modulus
is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$G = \frac{\partial P_s}{\partial \alpha_s} \tag{26–6}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres, the packing limit is about 0.63, which is the default value in ANSYS FLUENT. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

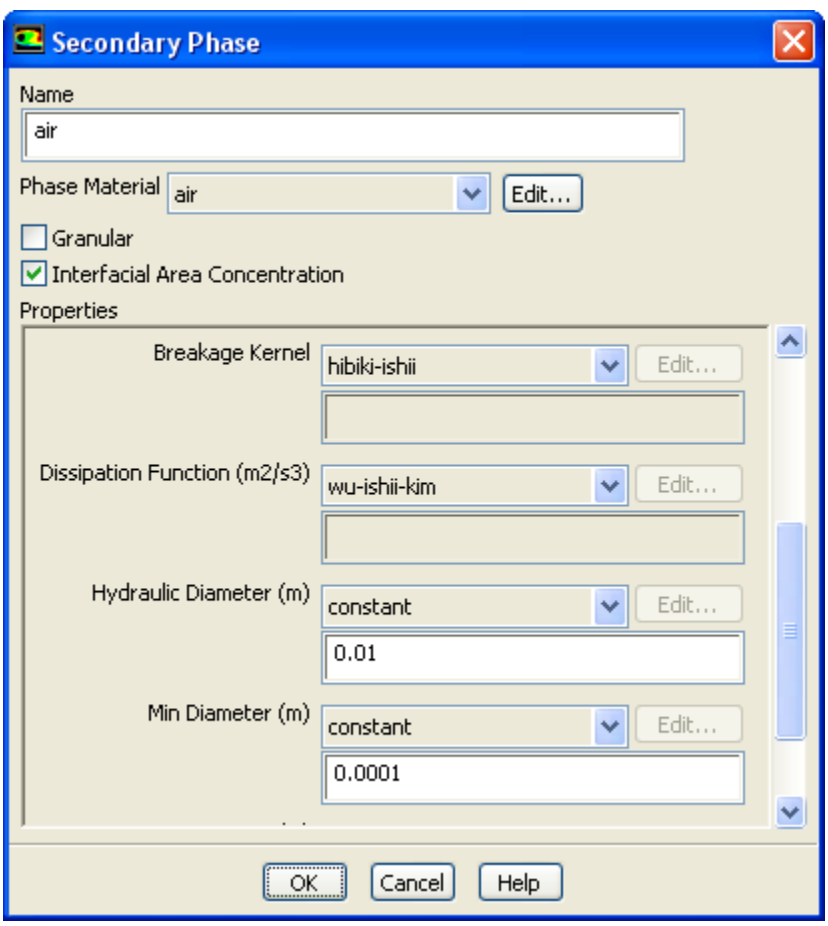
8. Click **OK** in the **Secondary Phase** dialog box.

26.4.1.4. Defining the Interfacial Area Concentration

To define the interfacial area concentration on the secondary phase in the mixture model, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the *Secondary Phase Dialog Box* (p. 1785) (*Figure 26.29* (p. 1149)).

Figure 26.29 The Secondary Phase Dialog Box Displaying the Interfacial Area Concentration Settings



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**.
6. Enable the **Interfacial Area Concentration** option. Make sure the **Granular** option is disabled for the **Interfacial Area Concentration** option to be visible in the interface.
7. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

   Diameter
   specifies the diameter of the particles or bubbles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the UDF Manual for details about user-defined functions. The **Diameter** recommended setting is **sauter-mean**, allowing for the effects of the interfacial area concentration values to be considered for mass, momentum and heat transfer across the interface between phases.

   Packing Limit
   specifies the maximum volume fraction for the particle/bubble phase.

   Surface Tension
   specifies the attractive forces between the interfaces. The surface tension for the liquid-air interface is set for both the **hibiki-ishii** and the **ishii-kim** models.

   Coalescence Kernel and Breakage Kernel
   allows you to specify the coalescence and breakage kernels. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, or **user-defined**. The two options, **hibiki-ishii** and **ishii-kim**, are described in detail in Interfacial Area Concentration in the Theory Guide.

   In addition to specifying the **hibiki-ishii** and **ishii-kim** as the coalescence and breakage kernels, you can also tune the properties of these two models by using the `/define/phases/iac-expert/hibiki-ishii-model` and `/define/phases/iac-expert/ishii-kim-model` text commands.

   For the Hibiki-Ishii model, you can specify the following parameters:

   Coefficient Gamma_c
   Coefficient K_c
   Coefficient Gamma_b
   Coefficient K_b
   alpha_max

   For the Ishii-Kim model, you can specify the following parameters:

   Coefficient Crc
   Coefficient Cwe
   Coefficient C
   Coefficient Cti
   alpha_max

   These values are discussed in greater detail in Interfacial Area Concentration in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dissipation Function

gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose amongst **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

The **wu-ishii-kim** option uses a simple algebraic correlation for ε:

$$\varepsilon = f_{TW}(1/2D_h)\, v_m^3 \qquad \textbf{(26–7)}$$

where

$$f_{TW} = \frac{0.316}{\left[(1-\alpha)Re_m\right]^{0.25}}$$

and

$$Re_m = \frac{\rho_m v_m D_h}{\mu_m}$$

where ρ_m, v_m, μ_m, and D_h are the mixture density, mixture velocity, mixture molecular viscosity, and hydraulic diameter of the flow path.

When you select the **wu-ishii-kim** model, you will set an additional input for **Hydraulic Diameter**.

Hydraulic Diameter

is the value used in *Equation 26–7* (p. 1151), should you use the **wu-ishii-kim** formulation.

Min/Max Diameter

are the limits of the bubble diameters.

26.4.1.5. Defining Drag Between Phases

For mixture multiphase flows with slip velocity, you can specify the drag function to be used in the calculation. The functions available here are a subset of those discussed in *Defining the Phases for the Eulerian Model* (p. 1154). See Relative (Slip) Velocity and the Drift Velocity in the Theory Guide for more information.

To specify drag laws, click **Interaction...** to open the *Phase Interaction Dialog Box* (p. 1791) (*Figure 26.30* (p. 1152)), and then click the **Drag** tab.

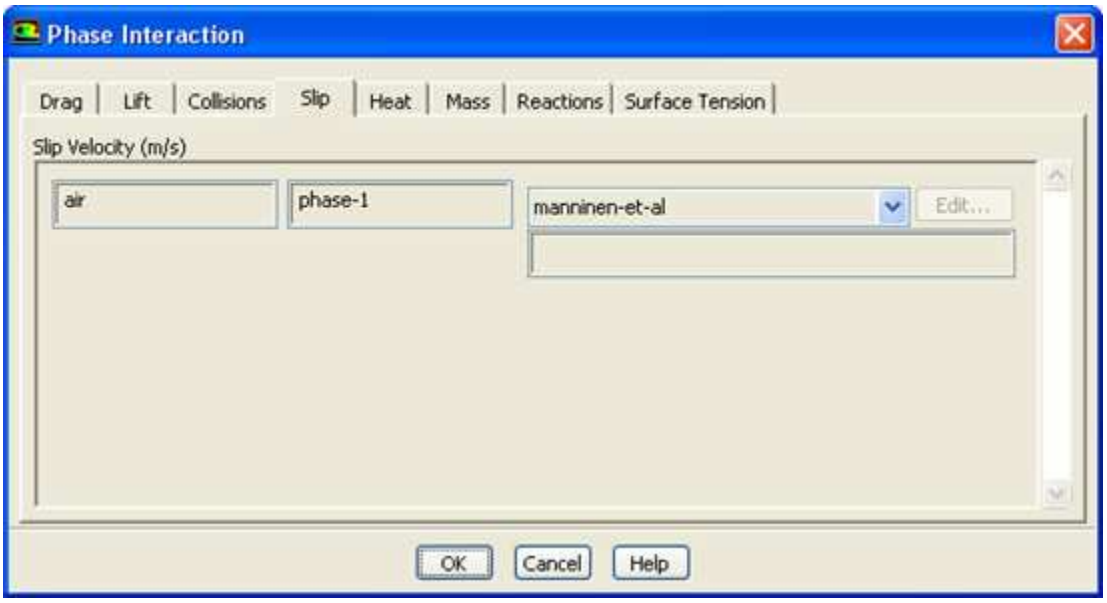
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.30 The Phase Interaction Dialog Box for the Mixture Model (Drag Tab)

26.4.1.6. Defining the Slip Velocity

If you are solving for slip velocities during the mixture calculation, and you want to modify the slip velocity definition, click **Interaction...** to open the *Phase Interaction Dialog Box* (p. 1791) (*Figure 26.31* (p. 1152)), and then click the **Slip** tab.

Figure 26.31 The Phase Interaction Dialog Box for the Mixture Model (Slip Tab)



Under **Slip Velocity**, you can specify the slip velocity function for each secondary phase with respect to the primary phase by choosing the appropriate item in the adjacent drop-down list.

- Select **maninnen-et-al** (the default) to use the algebraic slip method of Manninen et al. [*50*] (p. 2211), described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide.
- Select **none** if the secondary phase has the same velocity as the primary phase (i.e., no slip velocity).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **user-defined** to use a user-defined function for the slip velocity. See the UDF Manual for details.

26.4.2. Including Cavitation Effects

For mixture model calculations, it is possible to include the effects of cavitation, using ANSYS FLUENT's cavitation models described in Cavitation Models in the Theory Guide.

To enable the Singhal et al. cavitation model, use the `solve/set/expert` text command and answer `yes` to `use Singhal-et-al cavitation model?`. The **Singhal-Et-Al Cavitation Model** option will now be visible in the **Phase Interaction** dialog box, under the **Mass** tab. Enable this option to include the Singhalet al. cavitation model.

You will specify three parameters to be used in the calculation of mass transfer due to cavitation. Set the **Vaporization Pressure**, the **Surface Tension Coefficient**, and the **Non-Condensable Gas Mass Fraction**. The default value of p_{sat} is 3540 Pa, the vaporization pressure for water at ambient temperature. Note that p_{sat} and the surface tension are properties of the liquid, depending mainly on temperature. **Non-Condensable Gas Mass Fraction** is the mass fraction of dissolved gases, which depends on the purity of the liquid.

When multiple species are included in one or more secondary phases, or the heat transfer due to phase change needs to be taken into account, the mass transfer mechanism must be defined *before* activating the cavitation model. It may be noted, however, that for cavitation problems, at least *two* mass transfer mechanisms are defined:

- mass transfer from liquid to vapor.
- mass transfer from vapor to liquid.

To enable and set up the **Schnerr-Sauer** and **Zwart-Gerber-Belamri** cavitation models, refer to *Including Mass Transfer Effects* (p. 1104).

26.4.3. Modeling Compressible Flows

If you are using the mixture model for a compressible flow, note the following:

- Only one of the phases can be defined as a compressible ideal gas (i.e., you can select the ideal gas law for the density of only one phase's material). There is no limitation on using compressible liquids using user-defined functions.
- If you specify the total pressure at a boundary (e.g., for a pressure inlet or intake fan) the specified value for temperature at that boundary will be used as total temperature for the compressible phase, and as static temperature for the other phases (which are incompressible).
- For each mass flow inlet, you will need to specify mass flow or mass flux for each individual phase.

Important

Note that if you read a case file that was set up in a version of ANSYS FLUENT previous to 6.1, you will need to redefine the conditions at the mass flow inlets. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for more information on defining conditions for a mass flow inlet in mixture multiphase calculations.

See *Compressible Flows* (p. 503) for more information about compressible flows.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.5. Setting Up the Eulerian Model

For background information about the Eulerian model and the limitations that apply, refer to Overview of the Eulerian Model in the Theory Guide.

For additional information, please see the following sections:

26.5.1. Additional Guidelines for Eulerian Multiphase Simulations
26.5.2. Defining the Phases for the Eulerian Model
26.5.3. Setting Time-Dependent Parameters for the Explicit Volume Fraction Scheme
26.5.4. Modeling Turbulence
26.5.5. Including Heat Transfer Effects
26.5.6. Modeling Compressible Flows
26.5.7. Including the Dense Discrete Phase Model
26.5.8. Including the Boiling Model
26.5.9. Including the Immiscible Fluid Model

26.5.1. Additional Guidelines for Eulerian Multiphase Simulations

Once you have determined that the Eulerian multiphase model is appropriate for your problem (as described in Choosing a General Multiphase Model in the Theory Guide), you should consider the computational effort required to solve your multiphase problem. The required computational effort depends strongly on the number of transport equations being solved and the degree of coupling. For the Eulerian multiphase model, which has a large number of highly coupled transport equations, computational expense will be high. Before setting up your problem, try to reduce the problem statement to the simplest form possible.

Instead of trying to solve your multiphase flow in all of its complexity on your first solution attempt, you can start with simple approximations and work your way up to the final form of the problem definition. Some suggestions for simplifying a multiphase flow problem are listed below:

- Use a hexahedral or quadrilateral mesh (instead of a tetrahedral or triangular mesh).
- Reduce the number of phases.

You may find that even a very simple approximation will provide you with useful information about your problem.

See *Eulerian Model* (p. 1192) for more solution strategies for Eulerian multiphase calculations.

26.5.2. Defining the Phases for the Eulerian Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction for an Eulerian multiphase calculation are provided below.

26.5.2.1. Defining the Primary Phase

The procedure for defining the primary phase in an Eulerian multiphase calculation is the same as for a VOF calculation. See *Defining the Primary Phase* (p. 1136) for details.

26.5.2.2. Defining a Nongranular Secondary Phase

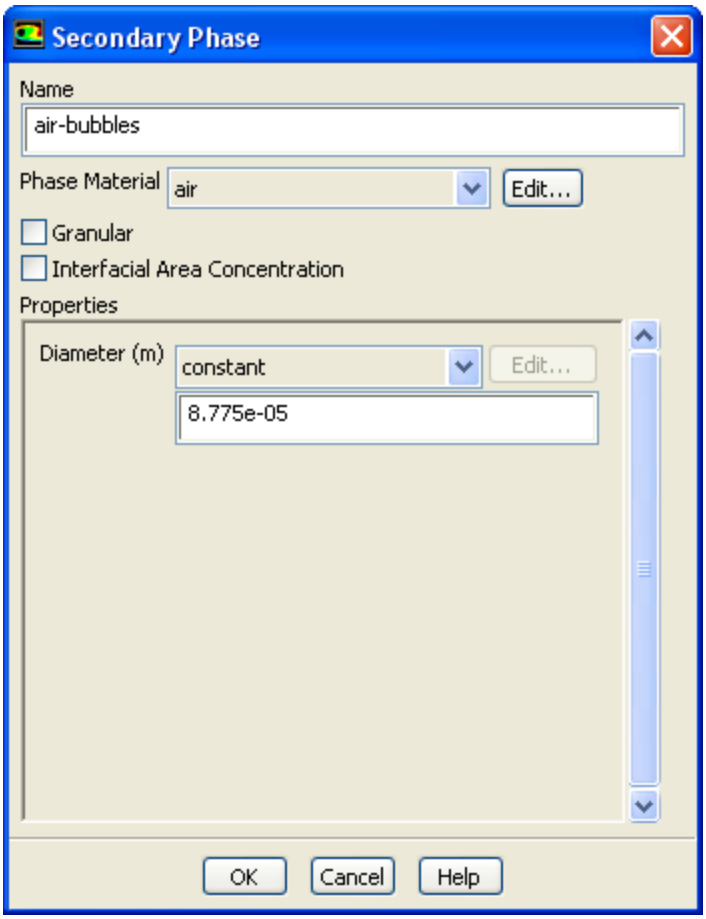
To define a nongranular (i.e., liquid or vapor) secondary phase in an Eulerian multiphase calculation, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Edit...** to open the *Secondary Phase Dialog Box* (p. 1785) (*Figure 26.32* (p. 1155)).

Figure 26.32 The Secondary Phase Dialog Box for a Nongranular Phase



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see *Defining the Primary Phase* (p. 1136)).
6. In the **Secondary Phase** dialog box, specify the **Diameter** of the bubbles or droplets of this phase. You can specify a constant value, or use a user-defined function. See the UDF Manual for details about user-defined functions.
7. Click **OK** in the **Secondary Phase** dialog box.

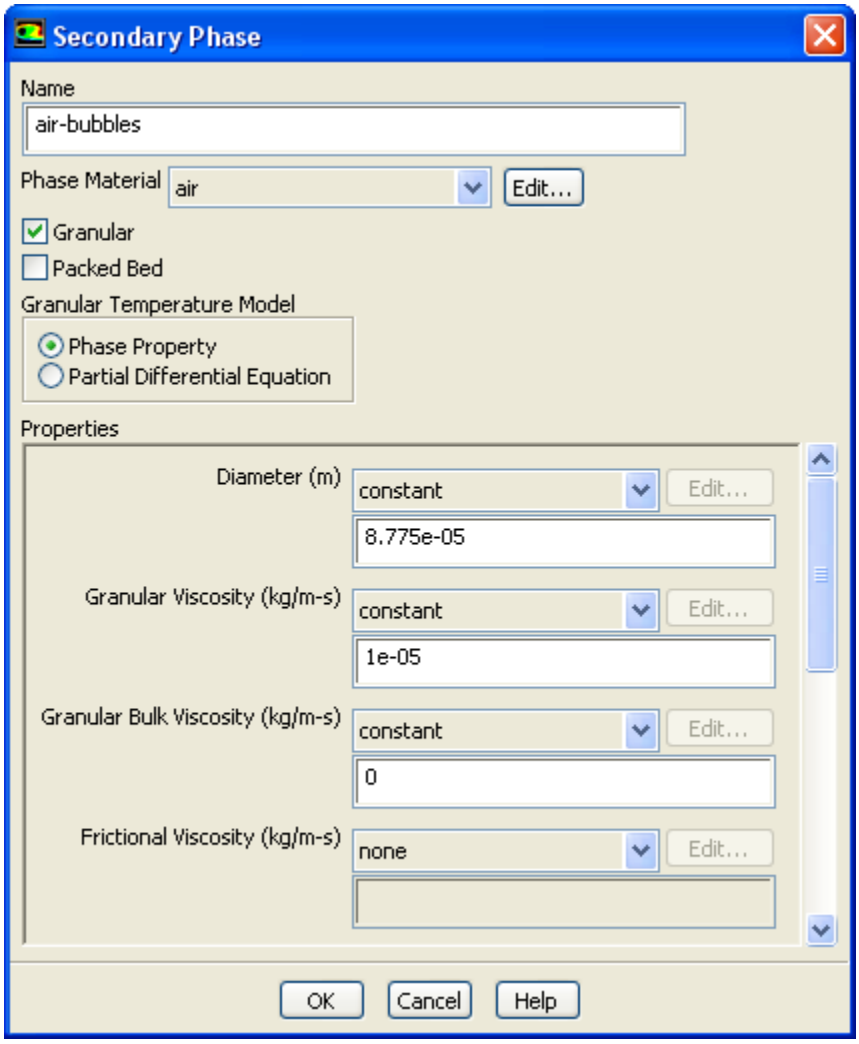
26.5.2.3. Defining a Granular Secondary Phase

To define a granular (i.e., particulate) secondary phase in an Eulerian multiphase calculation, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the *Secondary Phase Dialog Box* (p. 1785) (*Figure 26.33* (p. 1156)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.33 The Secondary Phase Dialog Box for a Granular Phase



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see *Defining the Primary Phase* (p. 1136)). For a granular phase (which must be placed in the fluid materials category, as mentioned in *Steps for Using a Multiphase Model* (p. 1091)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

Important

Note that all properties for granular flows can utilize user-defined functions (UDFs).

See the UDF Manual for details about user-defined functions.

6. Enable the **Granular** option.
7. (optional) Enable the **Packed Bed** option if you want to freeze the velocity field for the granular phase. Note that when you select the packed bed option for a phase, you should also use the fixed velocity option with a value of zero for all velocity components for all interior cell zones for that phase. Using

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fixed velocity in radial direction as a constant value (other than zero) does not ensure continuity. With the fixed velocity option, both velocity and pressure are fixed in a zone. Since face area in the radial direction is a function of radial distance from the axis, it will result in a mass conservation problem due to flux imbalance.

Using zero fixed velocity in the radial direction does not cause any problems related to mass conservation.

8. Specify the **Granular Temperature Model**. Choose either the default **Phase Property** option or the **Partial Differential Equation** option. See Granular Temperature in the Theory Guide for details.

9. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

Diameter

specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the UDF Manual for details about user-defined functions.

Granular Viscosity

specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 17–198 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 17–200 in the Theory Guide, select **gidaspow** to compute the value using Equation 17–201 in the Theory Guide, or select **user-defined** to use a user-defined function.

Granular Bulk Viscosity

specifies the solids bulk viscosity (λ_q in Equation 17–121 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 17–202 in the Theory Guide, or select **user-defined** to use a user-defined function.

Frictional Viscosity

specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 17–198 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 17–203 in the Theory Guide, select **johnson-et-al** to compute the value using Equation 17–208 in the Theory Guide, or select **user-defined** to use a user-defined function.

Angle of Internal Friction

specifies a constant value for the angle ϕ used in Schaeffer's expression for frictional viscosity (Equation 17–203 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer** or **user-defined** for the **Frictional Viscosity**.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17–208 in the Theory Guide, **syamlal-obrien** to apply Equation 17–148 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [*20*] (p. 2210). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the UDF manual for information on hooking a UDF.

Frictional Modulus

is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$G = \frac{\partial P_{friction}}{\partial \alpha_{friction}} \quad \textbf{(26–8)}$$

with $G \geq 0$, which is the **derived** option. You can also specify a **user-defined** function for the frictional modulus.

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5. This is only a general rule of thumb as there may be other factors involved.

Granular Conductivity

specifies the solids granular conductivity (k_{Θ_s} in Equation 17–211 in the Theory Guide). You can select **syamlal-obrien** to compute the value using Equation 17–212 in the Theory Guide, select **gidaspow** to compute the value using Equation 17–213 in the Theory Guide, or select **user-defined** to use a user-defined function. Note, however, that ANSYS FLUENT currently uses an algebraic relation for the granular temperature. This has been obtained by neglecting convection and diffusion in the transport equation, Equation 17–211 in the Theory Guide [*88*] (p. 2213).

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. Choose either the **algebraic**, the **constant**, or **user-defined** option.

Solids Pressure

specifies the pressure gradient term, $\nabla p_{s'}$ in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, **none**, or a **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastoopour**, or a **user-defined** option.

Elasticity Modulus

is defined as

$$G = \frac{\partial P_s}{\partial \alpha_s} \quad \textbf{(26–9)}$$

with $G \geq 0$.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres, the packing limit is about 0.63, which is the default value in ANSYS FLUENT. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

10. Click **OK** in the **Secondary Phase** dialog box.

26.5.2.4. Defining the Interfacial Area Concentration

To define the interfacial area concentration on the secondary phase in the Eulerian model, perform the following steps:

1. Select the phase (e.g., **phase-2**) in the **Phases** list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Edit...** to open the *Secondary Phase Dialog Box* (p. 1785) (*Figure 26.29* (p. 1149)).
3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**.
6. Enable the **Interfacial Area Concentration** option. Make sure the **Granular** option is disabled for the **Interfacial Area Concentration** option to be visible in the interface.
7. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

   Diameter
   specifies the diameter of the particles or bubbles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the UDF Manual for details about user-defined functions. The **Diameter** recommended setting is **sauter-mean**, allowing for the effects of the interfacial area concentration values to be considered for mass, momentum and heat transfer across the interface between phases.

   Packing Limit
   specifies the maximum volume fraction for the particle/bubble phase.

   Growth Rate
   allows you to specify the particle growth rate (m/s). You can select **none**, **constant**, or **user-defined** from the drop-down list. If you select **constant**, specify a value in the adjacent field. If you have a user-defined function (UDF) that you want to use to model the growth rate, you can choose the **user-defined** option and specify the appropriate UDF.

   Coalescence Kernel and Breakage Kernel
   allows you to specify the coalescence and breakage kernels. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, or **user-defined**. The two options, **hibiki-ishii** and **ishii-kim**, are described in detail in Interfacial Area Concentration in the Theory Guide.

   In addition to specifying the **hibiki-ishii** and **ishii-kim** as the coalescence and breakage kernels, you can also tune the properties of these two models by using the `/define/phases/iac-expert/hibiki-ishii-model` and `/define/phases/iac-expert/ishii-kim-model` text commands.

   For the Hibiki-Ishii model, you can specify the following parameters:

   Coefficient Gamma_c
   Coefficient K_c
   Coefficient Gamma_b
   Coefficient K_b
   alpha_max

   For the Ishii-Kim model, you can specify the following parameters:

   Coefficient Crc
   Coefficient Cwe
   Coefficient C
   Coefficient Cti
   alpha_max

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These values are discussed in greater detail in Interfacial Area Concentration in the Theory Guide.

Nucleation Rate
is a source term for the interfacial area concentration which models the rate of formation of the dispersed phase.

Min/Max Diameter
allow you to specify the minimum and maximum diameters of the secondary phase to which the interfacial area concentration model is applied, preventing some computing anomalies to spread beyond control.

26.5.2.5. Defining the Interaction Between Phases

For both granular and nongranular flows, you will need to specify the drag function to be used in the calculation of the momentum exchange coefficients. For granular flows, you will also need to specify the restitution coefficient(s) for particle collisions. It is also possible to include an optional lift force and/or virtual mass force (described below) for both granular and nongranular flows.

To specify these parameters, click **Interaction...** to open the *Phase Interaction Dialog Box* (p. 1791) and visit the **Drag**, **Collisions**, and **Lift** tabs.

Phases → **Interaction...**

26.5.2.5.1. Specifying the Drag Function

ANSYS FLUENT allows you to specify a drag function for each pair of phases. Perform the following steps:

1. Click the **Drag** tab.
2. For each pair of phases, select the appropriate drag function from the corresponding drop-down list.
   - Select **boiling-ishii** to use the fluid-fluid drag function described by Equation 17–284 in the Theory Guide. This option is available when the boiling model is enabled.
   - Select **schiller-naumann** to use the fluid-fluid drag function described by Equation 17–133 in the Theory Guide. The Schiller and Naumann model is the default method, and it is acceptable for general use in all fluid-fluid multiphase calculations.
   - Select **morsi-alexander** to use the fluid-fluid drag function described by Equation 17–137 in the Theory Guide. The Morsi and Alexander model is the most complete, adjusting the function definition frequently over a large range of Reynolds numbers, but calculations with this model may be less stable than with the other models.
   - Select **symmetric** to use the fluid-fluid drag function described by Equation 17–143 in the Theory Guide. The symmetric model is recommended for flows in which the secondary (dispersed) phase in one region of the domain becomes the primary (continuous) phase in another. For example, if air is injected into the bottom of a container filled halfway with water, the air is the dispersed phase in the bottom half of the container; in the top half of the container, the air is the continuous phase. The **symmetric** drag law is the default method for the **Immiscible Fluid Model**, which is available with Eulerian multiphase model.
   - Select **anisotropic** to use the fluid-fluid drag function described in Immiscible Fluid Model in the Theory Guide. The **anisotropic** drag law is recommended for free surface modeling. It is based on higher drag in the normal direction to the interface and lower drag in the tangenation direction to the interface. This is only available with **Immiscible Fluid Model**.
   - Select **universal-drag** for bubble-liquid and/or droplet-gas flow when the characteristic length of the flow domain is much greater than the averaged size of the particles. The universal drag law is

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

described using Equation 17–162 in the Theory Guide. When **universal-drag** is selected, you will need to set a value for the surface tension coefficient, under the **Surface Tension** tab, in the **Phase Interaction** dialog box. This value will apply to the primary phase and the secondary phase.

- Select **wen-yu** to use the fluid-solid drag function described by Equation 17–157 in the Theory Guide. The Wen and Yu model is applicable for dilute phase flows, in which the total secondary phase volume fraction is significantly lower than that of the primary phase.
- Select **gidaspow** to use the fluid-solid drag function described by Equation 17–159 in the Theory Guide. The Gidaspow model is recommended for dense fluidized beds.
- Select **syamlal-obrien** to use the fluid-solid drag function described by Equation 17–149 in the Theory Guide. The Syamlal-O'Brien model is recommended for use in conjunction with the Syamlal-O'Brien model for granular viscosity.
- Select **syamlal-obrien-symmetric** to use the solid-solid drag function described by Equation 17–161 in the Theory Guide. The symmetric Syamlal-O'Brien model is appropriate for a pair of solid phases.
- Select **constant** to specify a constant value for the drag function, and then specify the value in the text field.
- Select **user-defined** to use a user-defined function for the drag function (see the UDF Manual for details).
- If you want to temporarily ignore the interaction between two phases, select **none**.

26.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only)

For granular flows, you need to specify the coefficients of restitution for collisions between particles (e_{ls} in Equation 17–161 and e_{ss} in Equation 17–181 in the Theory Guide). In addition to specifying the restitution coefficient for collisions between each pair of granular phases, you will also specify the restitution coefficient for collisions between particles of the same phase.

Perform the following steps:

1. Click the **Collisions** tab to display the **Restitution Coefficient** inputs.
2. For each pair of phases, specify a constant restitution coefficient. All restitution coefficients are equal to 0.9 by default.

26.5.2.6. Including the Lift Force

For both granular and nongranular flows, it is possible to include the effect of lift forces ($\vec{F}_{lift}$ in Equation 17–123 in the Theory Guide) on the secondary phase particles, droplets, or bubbles. These lift forces act on a particle, droplet, or bubble mainly due to velocity gradients in the primary-phase flow field. In most cases, the lift force is insignificant compared to the drag force, so there is no reason to include it. If the lift force is significant (e.g., if the phases separate quickly), you may want to include this effect.

Important

Note that the lift force will be more significant for larger particles, but the ANSYS FLUENT model assumes that the particle diameter is much smaller than the interparticle spacing. Thus, the inclusion of lift forces is not appropriate for closely packed particles or for very small particles.

To include the effect of lift forces, perform the following steps:

1. Click the **Lift** tab to display the **Lift Coefficient** inputs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For each pair of phases, select the appropriate specification method from the corresponding drop-down list. Note that, since the lift forces for a particle, droplet, or bubble are due mainly to velocity gradients in the primary-phase flow field, you will not specify lift coefficients for pairs consisting of two secondary phases; lift coefficients are specified only for pairs consisting of a secondary phase and the primary phase.
   - Select **boiling-moraga** if you have the boiling model option enabled in the **Multiphase Model** dialog box. Information about this option is found in Interfacial Lift Force in the Theory Guide.
   - Select **boiling-tomiyama-et-al** if you have the boiling model option enabled in the **Multiphase Model** dialog box. Information about this option is found in Interfacial Lift Force in the Theory Guide.
   - Select **none** (the default) to ignore the effect of lift forces.
   - Select **constant** to specify a constant lift coefficient, and then specify the value in the text field.
   - Select **user-defined** to use a user-defined function for the lift coefficient (see the UDF Manual for details).

26.5.2.7. Including Surface Tension and Wall Adhesion Effects

As discussed in When Surface Tension Effects Are Important in the Theory Guide, the importance of surface tension effects depends on the value of the capillary number, Ca (defined by Equation 17–24 in the Theory Guide), or the Weber number, We (defined by Equation 17–25 in the Theory Guide). Surface tension effects can be neglected if Ca $\gg 1$ or We $\gg 1$.

Important

Note that the calculation of surface tension effects will be more accurate if you use a quadrilateral or hexahedral mesh in the area(s) of the computational domain where surface tension is significant. If you cannot use a quadrilateral or hexahedral mesh for the entire domain, then you should use a hybrid mesh, with quadrilaterals or hexahedra in the affected areas. ANSYS FLUENT also offers an option to use VOF gradients at the nodes for curvature calculations on meshes when more accuracy is desired. For more information, see Surface Tension and Adhesion in the Theory Guide.

If you want to include the effects of surface tension along the interface between one or more pairs of phases, as described in Surface Tension and Adhesion in the Theory Guide, click **Interaction...** to open the *Phase Interaction Dialog Box* (p. 1791) (*Figure 26.23* (p. 1139)).

Perform the following steps to include surface tension (and, if appropriate, wall adhesion and/or jump adhesion) effects along the interface between one or more pairs of phases:

1. Click the **Surface Tension** tab.
2. For each pair of phases between which you want to include the effects of surface tension, specify a constant surface tension coefficient. Alternatively you can specify a temperature dependent, polynomial, piece-wise polynomial, piecewise linear, or a user-defined surface tension coefficient. See Surface Tension and Adhesion in the Theory Guide for more information on surface tension, and the UDF Manual for more information on user-defined functions. All surface tension coefficients are equal to 0 by default, representing no surface tension effects along the interface between the two phases.
3. If you want to include wall adhesion, enable the **Wall Adhesion** option. When **Wall Adhesion** is enabled, you will need to specify the contact angle at each wall as a boundary condition (as described in *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.5.2.7.1. Including the Virtual Mass Force

For both granular and nongranular flows, it is possible to include the "virtual mass force" ($\vec{F}_{vm}$ in Equation 17–124 in the Theory Guide) that is present when a secondary phase accelerates relative to the primary phase. The virtual mass effect is significant when the secondary phase density is much smaller than the primary phase density (e.g., for a transient bubble column).

To include the effect of the virtual mass force, turn on the **Virtual Mass** option in the **Phase Interaction** dialog box. The virtual mass effect will be included for all secondary phases; it is not possible to enable it just for a particular phase.

26.5.3. Setting Time-Dependent Parameters for the Explicit Volume Fraction Scheme

If you are using the time-dependent volume fraction formulation in ANSYS FLUENT, an explicit solution for the volume fraction is obtained either once each time step or once each iteration, depending upon your inputs to the model. By default, ANSYS FLUENT will solve the volume fraction equation(s) once for each time step, except for the first time step. This means that the convective flux coefficients appearing in the other transport equations will not be completely updated each iteration, since the volume fraction fields will not change from iteration to iteration.

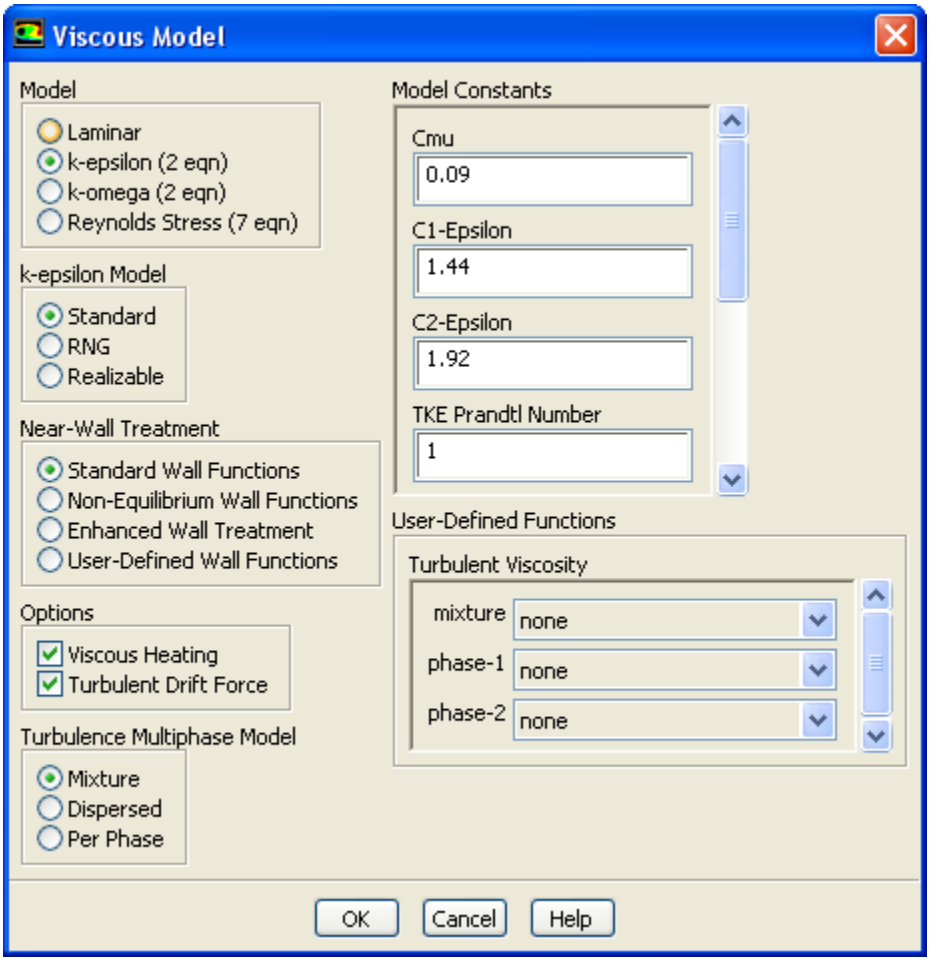
This formulation also applies to the VOF model, and is discussed in greater detail in *Setting Time-Dependent Parameters for the VOF Model* (p. 1143).

26.5.4. Modeling Turbulence

If you are using the Eulerian model to solve a turbulent flow, you will need to choose one of turbulence models described in Turbulence Models in the Theory Guide in the *Viscous Model Dialog Box* (p. 1645) (*Figure 26.34* (p. 1164)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.34 The Viscous Model Dialog Box for an Eulerian Multiphase Calculation



The procedure is as follows:

1. Select **k-epsilon**, **k-omega**, or **Reynolds Stress** under **Model**.
2. Select the desired **k-epsilon Model**, **k-omega Model**, or **Reynolds-Stress Model** and any other related parameters, as described for single-phase calculations in *Steps in Using a Turbulence Model* (p. 655).
3. Under **Turbulence Multiphase Model** or **RSM Multiphase Model**, indicate the desired multiphase turbulence model (see Turbulence Models in the Theory Guide for details about each):
   - Select **Mixture** to use the mixture turbulence model. This is the default model.
   - Select **Dispersed** to use the dispersed turbulence model. This model is applicable when there is clearly one primary continuous phase and the rest are dispersed dilute secondary phases.
   - Select **Per Phase** to use a k- ε or k- ω turbulence model for each phase. This model is appropriate when the turbulence transfer among the phases plays a dominant role.

You can include the effect of drift velocity by enabling **Turbulent Drift Force** under **Options**.

Important

This option is only available if either the **Eulerian** multiphase model is selected, or the **Mixture** model with **Slip Velocity** is enabled in the **Multiphase Model** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, you can also allow for drift velocity by performing the following:

1. If it is not already done, set the **Turbulence Multiphase Model** to **Mixture**, **Dispersed**, or **Per Phase** in the **Viscous Model** dialog box.
2. Enter the `multiphase-options` text command in the console.

   `define` → `models` → `viscous` → `multiphase-turbulence` → `multiphase-options`

   ```
   /define/models/viscous/multiphase-turbulence> multiphase-options
   Enable dispersion force in momentum? [no] yes
   Enable interphase turbulence source? [no] yes
   ```

   The effect of the drift velocity is influenced both by the momentum equation and, to a lesser extent, the turbulence equation. Therefore, you should answer `yes` to both questions to take into account the effect of drift velocity.

   Important

   Anisotropic drag applicable to free surface modeling is only compatible with the **Mixture** turbulence model.

26.5.4.1. Including Source Terms

By default, the interphase momentum, k, and ε sources are not included in the calculation. If you want to include any of these source terms, you can enable them using the `multiphase-options` command in the `define/models/viscous/multiphase-turbulence/` text menu. Note that the inclusion of these terms can slow down convergence noticeably. If you are looking for additional accuracy, you may want to compute a solution first without these sources, and then continue the calculation with these terms included. In most cases these terms can be neglected.

26.5.4.2. Customizing the k- ε Multiphase Turbulent Viscosity

If you are using the k- ε multiphase turbulence model, a user-defined function can be used to customize the turbulent viscosity for each phase. This option will enable you to modify μ_t in the k- ε model. For more information, see the UDF Manual.

In the **Viscous Model** dialog box, under **User-Defined Functions**, select the appropriate user-defined function in the **Turbulent Viscosity** drop-down list.

26.5.5. Including Heat Transfer Effects

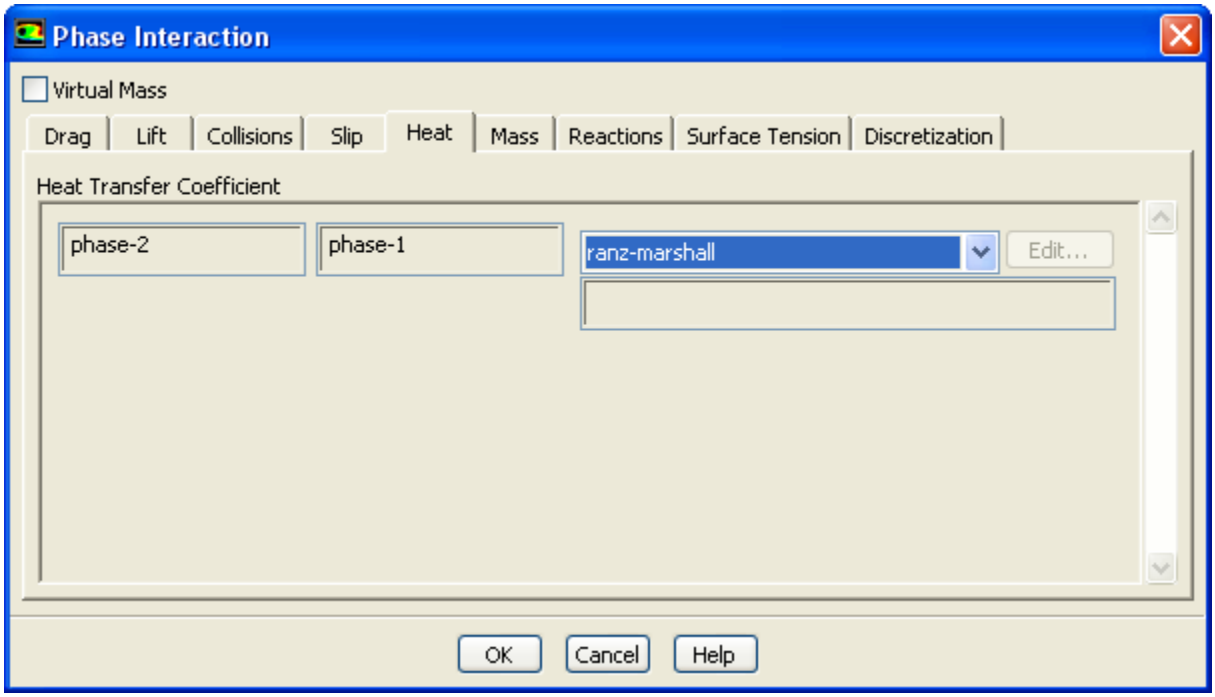
To define heat transfer in a multiphase Eulerian simulation, you will need to visit the **Phase Interaction** dialog box, after you have enabled the energy equation in the **Energy** dialog box.

Phases → **Interaction...**

1. Click the **Interaction...** button to open the **Phase Interaction** dialog box (e.g., *Figure 26.35* (p. 1166)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.35 The Phase Interaction Dialog Box for Heat Transfer



2. Click on the **Heat** tab in the **Phase Interaction** dialog box.
3. Select the desired correlation for the **Heat Transfer Coefficient**. Note the following regarding the available choices:

   gunn
   is frequently used for Eulerian multiphase simulations involving a granular phase.

   ranz-marshall
   is frequently used for Eulerian multiphase simulations not involving a granular phase.

   none
   allows you to ignore the effects of heat transfer between the two phases

   user-defined
   allows you to implement a correlation reflecting a model of your choice, through a user-defined function.
4. Set the appropriate thermal boundary conditions. You will specify the thermal boundary conditions for each individual phase on most boundaries, and for the mixture on some boundaries. See *Cell Zone and Boundary Conditions* (p. 205) for more information on boundary conditions, and *Eulerian Model* (p. 1111) for more information on specifying boundary conditions for a Eulerian multiphase calculation.

See Description of Heat Transfer in the Theory Guide for more information on heat transfer in the framework of a Eulerian multiphase simulation.

26.5.6. Modeling Compressible Flows

You can model compressible multiphase flows, and can use it in conjunction with the energy multiphase equations and available multiphase turbulence models. When using the Eulerian multiphase model for a compressible flow, note the following:

- While you can specify both compressible gas phases and compressible liquid phases, you can only define one of the phases as a compressible ideal gas (i.e., you can select the **ideal-gas** for the density

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Create/Edit Materials** dialog box of only one phase's material). There is no limitation on using compressible liquids using user-defined functions.

- You can define only one compressible fluid phase.
- For each mass flow inlet, you will need to specify mass flow or mass flux for each individual phase.
- If you specify the total pressure at a boundary (e.g., for a pressure inlet or intake fan), ANSYS FLUENT will use the specified value for temperature at that boundary as total temperature for the compressible phase, and as static temperature for the other phases (which are incompressible).

Important

Note that if you read a case file that was set up in a version of ANSYS FLUENT previous to 6.1, you will need to redefine the conditions at the mass flow inlets. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for more information on defining conditions for a mass flow inlet in Eulerian multiphase calculations.

See *Compressible Flows* (p. 503) for more information about compressible flows.

26.5.7. Including the Dense Discrete Phase Model

If you are using the Eulerian multiphase model (*Setting Up the Eulerian Model* (p. 1154)), you have the option of including the **Dense Discrete Phase Model** (Dense Discrete Phase Model in the Theory Guide).

Important

- This model is only available with the **Eulerian** multiphase model.
- Enabling this model automatically enables the DPM model. You will notice that **Interaction with Continuous Phase** in the **Discrete Phase Model** dialog box is enabled.

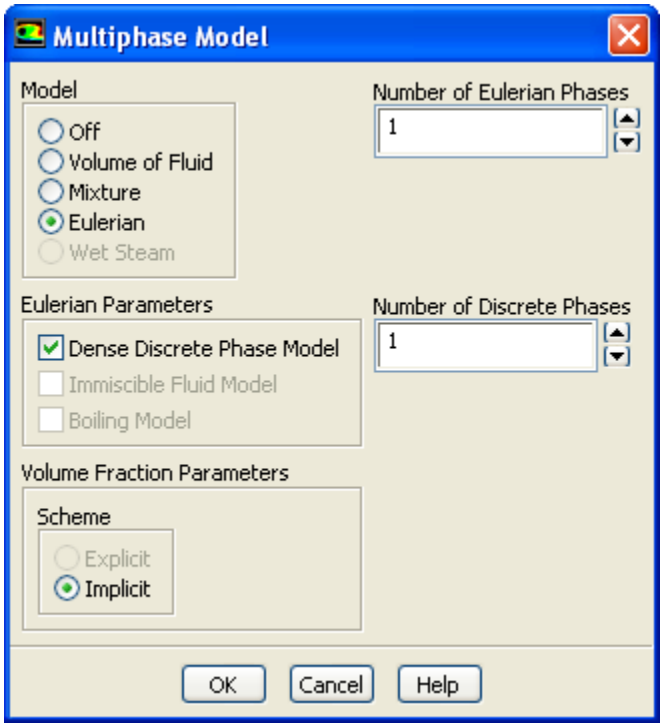
The required work flow when using the dense discrete phase model is as follows:

1. Set up the **Multiphase Model** dialog box (*Figure 26.36* (p. 1168)) to include the dense discrete phase model parameters.

   Models → **Multiphase** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.36 The Dense Discrete Phase Model



a. Enable **Dense Discrete Phase Model** under **Eulerian Parameters**.

b. Set the **Number of Discrete Phases** that are present in your case.

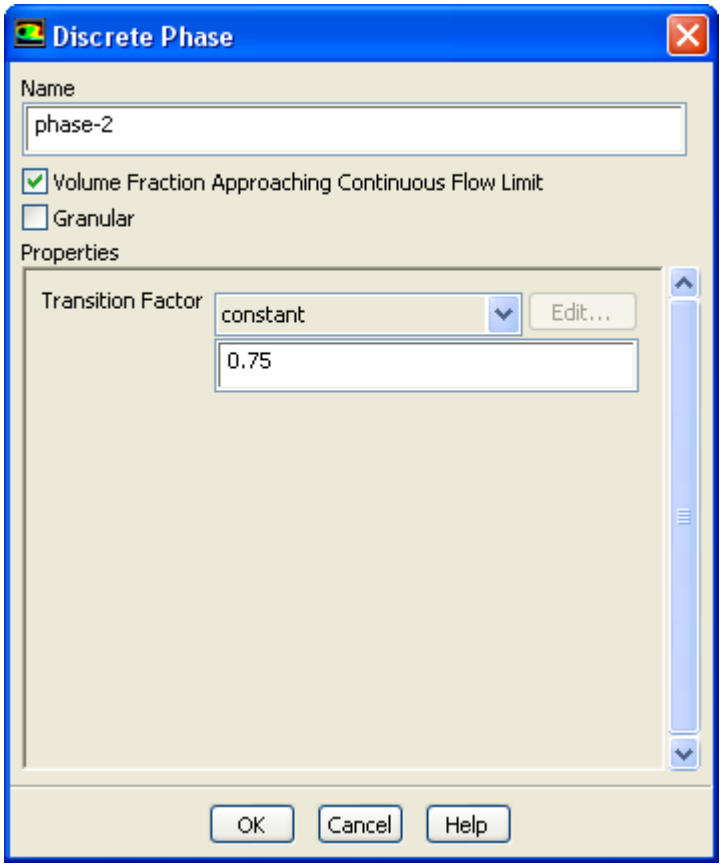
2. Open the **Phases** task page, in order to define the phases.

Phases

a. Define the discrete phase, by selecting the phase from the **Phases** selection list that is labeled **Discrete Phase** and clicking the **Edit...** button. Then set up the properties in the **Discrete Phase** dialog box that opens, as shown in *Figure 26.37* (p. 1169).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.37 The Discrete Phase Dialog Box



Note that for nongranular flow, no additional inputs are required here (since the material is set automatically in the background, and the diameter is part of the solution).

b. Enable **Volume Fraction Approaching Continuous Flow Limit** to specify a **Transition Factor**. The default value is assumed to be 0.75, which corresponds to the closest sphere packing for monosized spheres (a factor of 4/3). The transition volume fraction here will be 0.5625 ($0.75 * 4/3$). Note that this factor is not limited to a range between 0 and 1. In other words, you can specify values outside this range. For example, a value of 1.2 will give a transition volume fraction of 0.9. You can also specify locally variable values using the `DEFINE_PROPERTY` user-defined macro, depending on the local particle size distribution, as an example.

   See DEFINE_PROPERTY UDFs in the UDF Manual for information about the `DEFINE_PROPERTY`.

c. Define the primary and secondary phases, as described in *Defining the Phases for the Eulerian Model* (p. 1154).

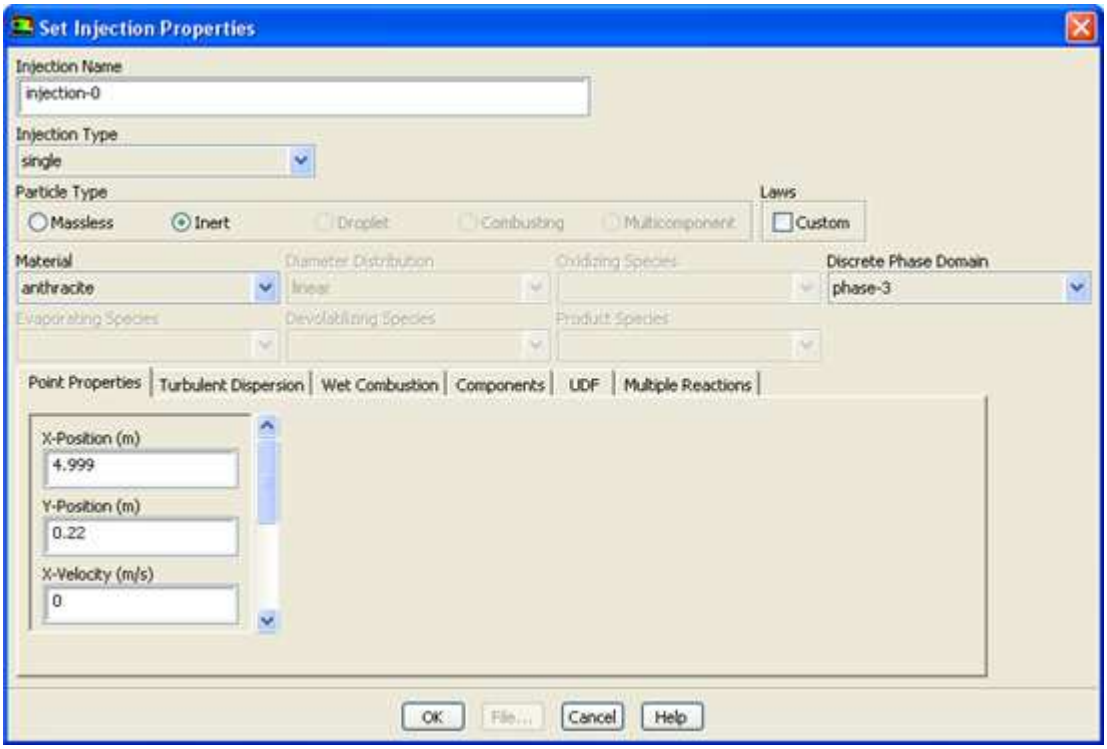
3. Define the injections using the **Injections** dialog box.

   Define → **Injections...**

   a. Create a new injection by clicking the **Create** button in the **Injections** dialog box, or edit an existing injection by selecting the injection from the **Injections** list and clicking the **Set...** button. The **Set Injection Properties** dialog box will open (*Figure 26.38* (p. 1170)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.38 The Set Injection Properties Dialog Box



b. Select the discrete phase from the **Discrete Phase Domain** drop-down list.

4. Define the material properties for each injection.

Materials

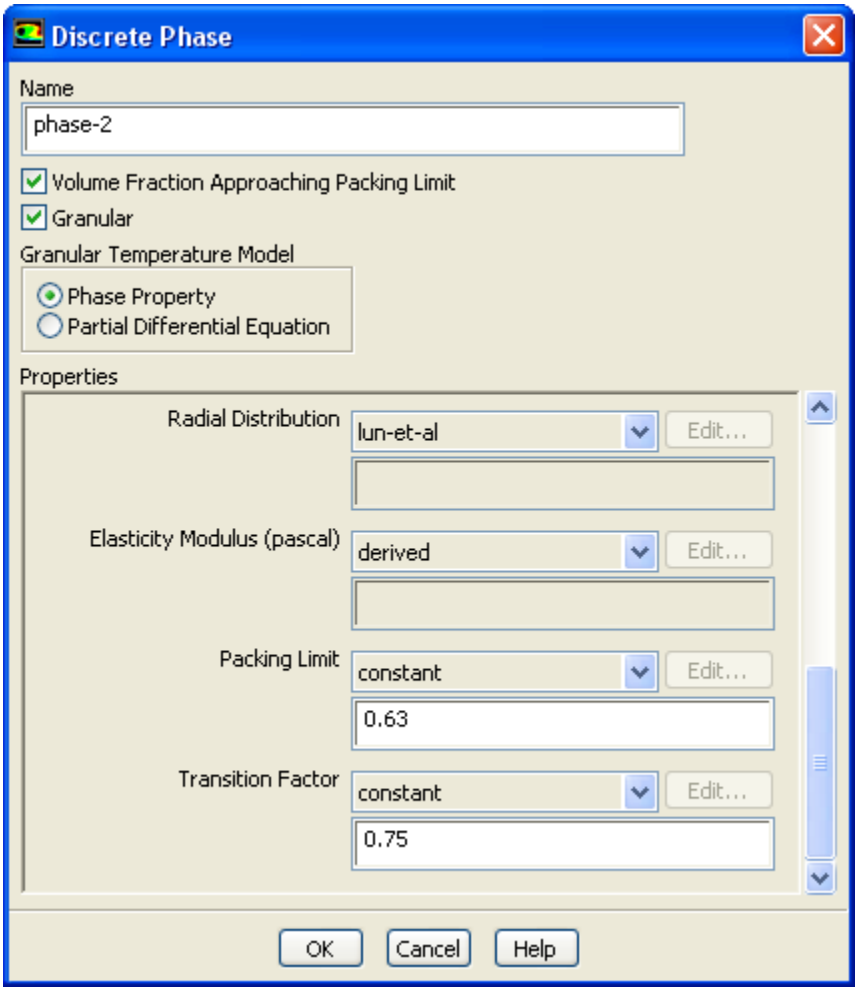
26.5.7.1. Defining a Granular Discrete Phase

To define a granular (i.e., particulate) discrete phase in an Eulerian multiphase calculation, perform the following steps:

1. Select the phase (e.g., **Discrete Phase**) in the **Phases** list.
2. Click **Edit...** to open the **Discrete Phase** dialog box (*Figure 26.39* (p. 1171)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.39 The Discrete Phase Dialog Box for a Granular Phase



3. In the **Discrete Phase** dialog box, enter a **Name** for the phase.
4. Enable the **Granular** option.
5. Enable **Volume Fraction Approaching Packing Limit** to prevent the unlimited accumulation of particles, which are operating at packing limit conditions.
6. Specify the **Transition Factor** as either a **constant** or a **user-defined** function. The default value for the **Transition Factor** is 0.75. The transition criterion is based on the local particle volume fraction of the given discrete phase and is specified as a factor multiplied by the maximum packing limit (also a user specified value). In other words, For a typical granular phase with a maximum packing limit of 0.63, the transition volume fraction is the product of 0.75 and 0.63 which is equal to 0.4725.

You can now define all other fields of the discrete phase in a similar manner to that described in *Defining a Granular Secondary Phase* (p. 1155).

26.5.8. Including the Boiling Model

If you are using the Eulerian multiphase model (*Setting Up the Eulerian Model* (p. 1154)), you have the option of including the **Boiling Model** (see Wall Boiling Models in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

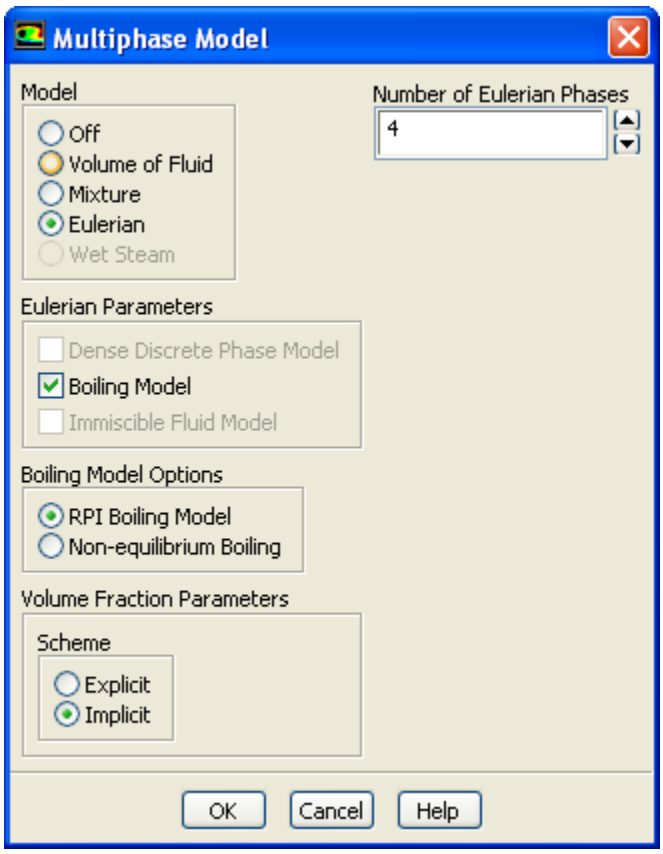
- This model is only available with the **Eulerian** multiphase model.
- This model is only available with the **Pressure-Based** solver.

The required work flow when using the boiling model is as follows:

1. Open the **Multiphase Model** dialog box (*Figure 26.40* (p. 1172)).

   Models → **Multiphase** → **Edit...**

   Figure 26.40 The Boiling Model



   a. Enable **Boiling Model** under **Eulerian Parameters**.

   b. Select either **RPI Boiling Model** or **Non-equilibrium Boiling** as boiling model options. Information about the two options is available in RPI Model and Non-equilibrium Subcooled Boiling in the Theory Guide.

   c. Set the total **Number of Eulerian Phases** that are present in your case. This can consist of two phases: liquid and vapor, which are directly involved in boiling mass transfer; or it can include "non-boiling phases" or "species". If a system consists of three phases: liquid, vapor, and air, then the **Number of Eulerian Phases** will be 3, where air is the non-boiling phase.

2. Make sure the energy option is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

The energy option will be automatically turned on when the boiling model is enabled.

Models → **Energy** → **Edit...**

3. Choose one of the turbulence models that is available with the Eulerian multiphase model and enable **Turbulent Drift Force**.

Models → **Viscous** → **Edit...**

Important

The turbulent drift force is an important force in the boiling model and should generally be included.

4. Open the **Create/Edit Materials** dialog box and specify the material properties for the liquid, vapor, solid, and any other phases that may exist.

Materials → **Fluid** → **Create/Edit...**

Important

For the liquid and vapor phases, the **Standard State Enthalpy** must be specified, as it is used in the computation of the **Latent Heat**.

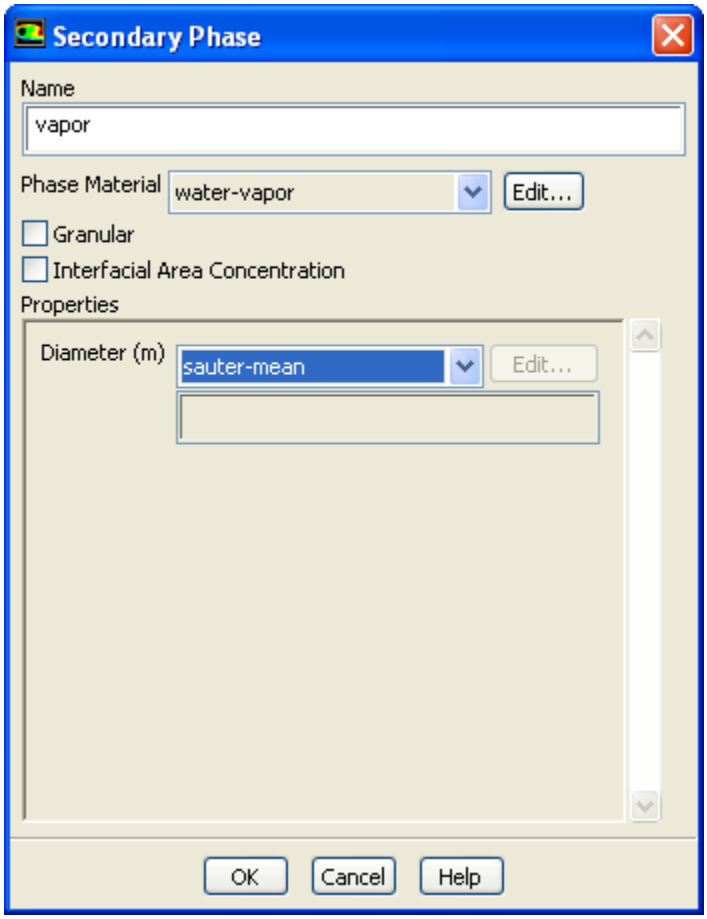
5. Open the **Phases** task page, in order to define the phases.

Phases

Define the liquid as the primary phase by selecting the phase from the **Phases** selection list that is labeled **Primary Phase** and clicking the **Edit...** button. Define the vapor as the secondary phase. In the **Secondary Phase** dialog box, the **Diameter** is set to **sauter-mean** by default, but you have the option of setting it up as a **constant** value or a **user-defined** function, as shown in *Figure 26.41* (p. 1174).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.41 The Secondary Phase Dialog Box



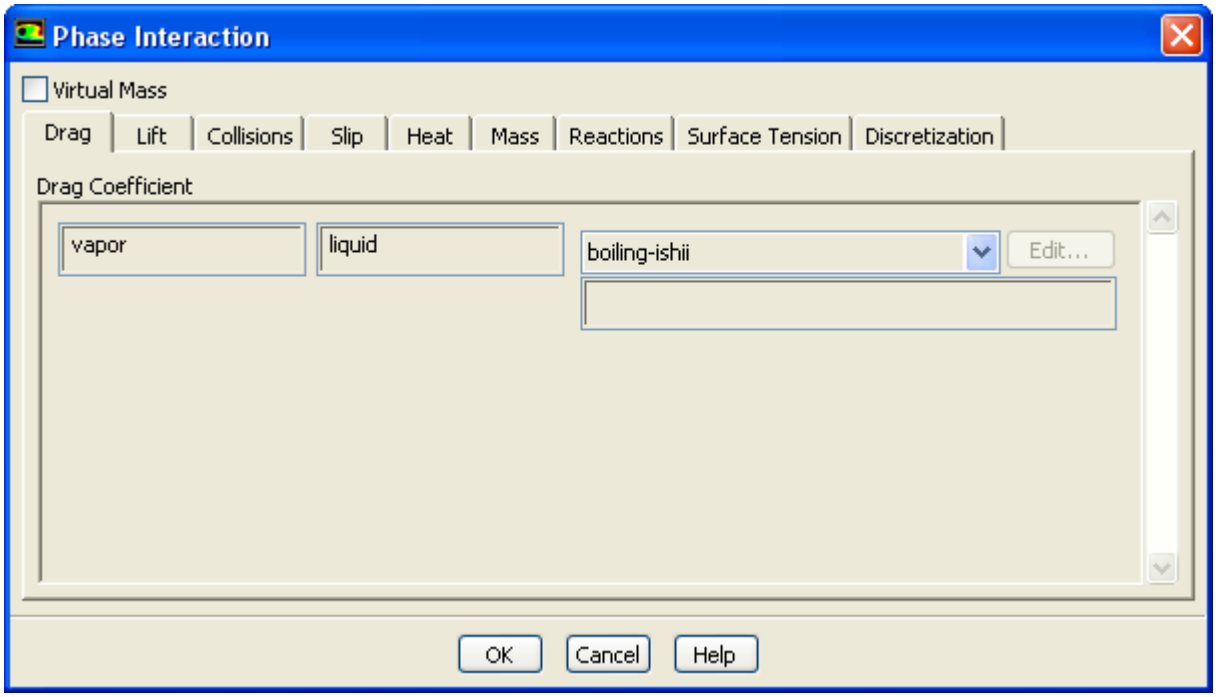
6. Open the **Phase Interaction** dialog box.

Phases → **Liquid** → **Interaction...**

a. In the **Drag** tab, you have a choice of **boiling-ishii**, **universal-drag**, **schiller-naumann**, **symmetric**, **morsi-alexander**, or **user-defined**. See *Specifying the Drag Function* (p. 1160) for definitions of the drag options.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.42 The Phase Interaction Dialog Box

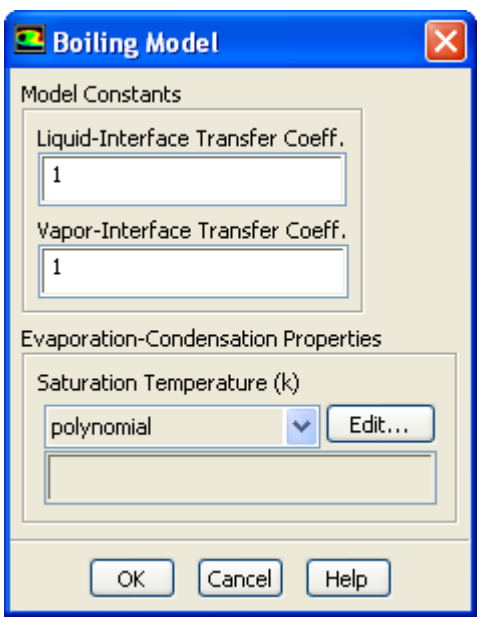


b. In the **Lift** tab, select **boiling-moraga**, **boiling-tomiyama-et-al**, or **user-defined**.

c. In the **Heat** tab, select **ranz-marshall** or **user-defined**.

d. In the **Mass** tab, set the **Number of Mass Transfer Mechanisms** to 1 and make sure the transfer is always from the liquid to the vapor phase. Select **boiling** under **Mechanism**. The Boiling Model dialog box will open (*Figure 26.43* (p. 1175)), where you will set the transfer coefficients and the saturation temperature. The default values for the liquid and vapor interface transfer coefficients are 1.

Figure 26.43 The Boiling Model Dialog Box



e. In the **Surface Tension** tab, select **constant** and enter the desired value.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Enable **Gravity** and set the **Operating Pressure** in the **Operating Conditions** dialog box.

   Important

   Make sure gravity is included when the boiling model is used.

8. Set up the conditions at inlets, outlets, and thermal conditions for walls.

   Boundary Conditions

   For the quenching wall heat flux, Koncar et al [*114*] (p. 2215) have suggested that in order to avoid grid dependence when calculating the quenching heat transfer, a factor that relates the temperature at a fixed normalized distance (y+ = 250) to the temperature at the near wall cell needs to be applied. Please contact a technical support engineer for more information.

   The wall boiling models are compatible with three different wall boundaries: isothermal wall, specified heat flux, and specified heat transfer coefficient (coupled wall boundary).

9. Choose either **Multiphase Coupled** or **Full Multiphase Coupled** as the pressure-velocity coupling scheme.

   Solution Methods

   Note

   Although **Phase Coupled SIMPLE** is also available, it is generally less robust and is not recommended for use in steady cases involving boiling or mass transfer.

10. The following solution strategies are recommended for boiling model simulations:

    Solution Controls

    - **Courant Number**: use a value between 1 and 20 is recommended. As a first try, use a value of 10.
    - **Explicit Relaxation Factors**: use the default values of 1 for **Momentum** and **Pressure**.
    - **Vaporization Mass**: use an under-relaxation factor between 0.5 and 1 is recommended. As a first try, use a value of 1.
    - **Volume Fraction**: use an under-relaxation factor between 0.3 and 0.5 is recommended.
    - **Turbulent Kinetic Energy** : use an under-relaxation factor between 0.3 and 0.8 is recommended.
    - **Turbulent Viscosity** : use an under-relaxation factor between 0.5 and 1.0 is recommended.
    - **Energy** : use an under-relaxation factor between 0.5 and 0.8 is recommended.

26.5.9. Including the Immiscible Fluid Model

After you have selected the Eulerian multiphase model (*Setting Up the Eulerian Model* (p. 1154)), you can enable the **Immiscible Fluid Model** (Immiscible Fluid Model in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

- This model is only available with the **Eulerian** multiphase model.
- You cannot use this model with the **Dense Discrete Phase Model**.
- After selecting this model, VOF sharpening schemes such as **Geo-Reconstruct** and **CICSAM** become available for the Eulerian multiphase model, as well as the **Compressive** discretization scheme. The default scheme with this model is **Geo-Reconstruct**.
- You can use the following drag laws with this model: **symmetric**, **anisotropic-drag**, **user-defined**, and **none**. The Default drag law with this model is **symmetric**.
- The anisotropic drag law is only compatible with the Eulerian multiphase model.
- Anisotropic drag applicable to free surface modeling is only compatible with the **Mixture** turbulence model.
- When including turbulence, only the mixture turbulence model can be used with anisotropic drag.

Important

The immiscible fluid model for the Eulerian multiphase allows you to use the sharpening schemes Geo-Reconstruct, compressive, and CICSAM with the **Explicit** VOF option. This model should be enabled only for the cases requiring sharp interface treatment between phases.

If you want to use the anisotropic drag law, perform the following steps:

1. Define the drag law in the **Phase Interaction** dialog box.

   Phases → **Interaction...**

2. Click the **Drag** tab to display the **Drag Coefficient** inputs.
3. For each pair of phases, select the appropriate drag law from the corresponding drop-down list.
   - Select **anisotropic-drag** when there is higher drag in the normal direction to the interface and lower drag in the tangential direction to the interface. For details about this drag law, refer to Immiscible Fluid Model in the Theory Guide.

     To specify the input parameters for anisotropic drag, you will need to use the `/solve/set/mp-mfluid-aniso-drag` text command. The options for an `Anisotropic Drag Method` of `0` (which is based on the symmetric drag), are as follows:

     ```
     Anisotropic Drag Method
       [0]

     Normal Interfacial Drag Friction Factor
       [1000000]

     Tangential Interfacial Drag Friction Factor
       [10]

     Length scale
       [0.0001]
     ```

     The options for an `Anisotropic Drag Method` of 1 are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Anisotropic Drag Method
   [0] 1

Viscosity option
   [2]

Normal Interfacial Drag Friction Factor
   [1000000]

Tangential Interfacial Drag Friction Factor
   [10]

Length scale
   [0.0001]
```

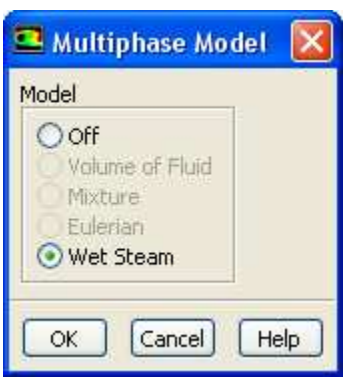
4. Click the **Discretization** tab to enable the use of the **Phase Localized Compressive Scheme**. For information about applying the various schemes, please refer to *Discretizing Using the Phase Localized Compressive Scheme* (p. 1141).

26.6. Setting Up the Wet Steam Model

After you have enabled the density-based solver in ANSYS FLUENT, you can activate the wet steam model (see Wet Steam Model Theory in the Theory Guide) by opening the **Multiphase Model** dialog box and selecting the **Wet Steam** option.

Models → **Multiphase** → **Edit...**

Figure 26.44 The Multiphase Model Dialog Box with the Wet Steam Model Activated



This section includes information about using your own property functions and data with the wet steam model. Solution settings and strategies for the wet steam model can be found in *Wet Steam Model* (p. 1195). Postprocessing variables are described in *Model-Specific Variables* (p. 1196).

For additional information, please see the following sections:

26.6.1. Using User-Defined Thermodynamic Wet Steam Properties
26.6.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF)
26.6.3. Compiling Your UDWSPF and Building a Shared Library File
26.6.4. Loading the UDWSPF Shared Library File
26.6.5. UDWSPF Example

26.6.1. Using User-Defined Thermodynamic Wet Steam Properties

ANSYS FLUENT allows you to use your own property functions and data with the wet steam model. This is achieved with user-defined wet steam property functions (UDWSPF).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These user-defined functions are written in the C programming language and there is a certain programming format that must be used so that you can build a successful library that can be loaded into the ANSYS FLUENT code.

The following is the procedure for using the user-defined wet steam property functions (UDWSPF):

1. Define the wet steam equation of state and all related thermodynamic and transport property equations.
2. Create a C source code file that conforms to the format defined in this section.
3. Start ANSYS FLUENT and set up your case file in the usual way.
4. Turn on the wet steam model.
5. Compile your UDWSPF C functions and build a shared library file using the text user interface.

   `define` → `models` → `multiphase` → `wet-steam` → `compile-user-defined-wetsteam-functions`
6. Load your newly created UDWSPF library using the text user interface.

   `define` → `models` → `multiphase` → `wet-steam` → `load-unload-user-defined-wet-steam-library`
7. Run your calculation.

Important

Note that the UDWSPF can only be used when the wet steam model is activated. Therefore, the UDWSPF are available for use with the density-based solver only.

26.6.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF)

Creating a UDWSPF C function library is reasonably straightforward:

- The code must contain the `udf.h` file inclusion directive at the beginning of the source code. This allows the definitions for `DEFINE` macros and other ANSYS FLUENT functions to be accessible during the compilation process.
- The code must include at least one in the UDF's `DEFINE` functions (i.e. `DEFINE_ON_DEMAND`) to be able to use the compiled UDFs utility.
- Any values that are passed to the solver by the UDWSPF or returned by the solver to the UDWSPF are assumed to be in SI units.
- You must use the principle set of user-defined wet steam property functions in your UDWSPF library, as described in the list that follows. These functions are the mechanism by which your thermodynamic property data is transferred to the ANSYS FLUENT solver.

The following lists the user-defined wet steam property function names and arguments, as well as a short description of their functions. Function inputs from the ANSYS FLUENT solver consist of one or more of the following variables: T = temperature (K), P = pressure (Pa), and ρ = vapor-phase density (kg/m^3).

- `void wetst_init(Domain *domain)`

  This will be called when you load the UDWSPF. You use it to initialize wet steam model constants or your own model constants. It returns nothing.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `real wetst_satP(real T)`

  This is the saturated pressure function, which takes on temperature in K and returns saturation pressure in Pa.

- `real wetst_satT(real P, real T)`

  This is the saturated temperature function, which takes on pressure in Pa and a starting guess temperature in K and returns saturation temperature in K.

- `real wetst_eosP(real rho, real T)`

  This is the equation of state, which takes on vapor density in kg/m^3 and Temperature in K and returns pressure in Pa.

- `real wetst_eosRHO(real P, real T)`

  This is the equation of state, which takes on pressure in Pa and temperature in K and returns vapor density in kg/m^3.

- `real wetst_cpv(real T, real rho)`

  This is the vapor specific heat at constant pressure, which takes on temperature in K and vapor density in kg/m^3 and returns specific heat at constant pressure in J/kg/K.

- `real wetst_cvv(real T, real rho)`

  This is the vapor specific heat at constant volume, which takes on temperature in K and vapor density in kg/m^3 and returns specific heat at constant volume in J/kg/K.

- `real wetst_hv(real T,real rho)`

  This is the vapor specific enthalpy, which takes on temperature in K and vapor density in kg/m^3 and returns specific enthalpy in J/Kg.

- `real wetst_sv(real T, real rho)`

  This is the vapor specific entropy, which takes on temperature in K and vapor density in kg/m^3 and returns specific entropy in J/Kg/K.

- `real wetst_muv(real T, real rho)`

  This is the vapor dynamic viscosity, which takes on temperature in K and vapor density in kg/m^3 and returns viscosity in kg/m/s.

- `real wetst_ktv(real T, real rho)`

  This is the vapor thermal conductivity, which takes on temperature in K and vapor density in kg/m^3 and returns thermal conductivity in W/m/K.

- `real wetst_rhol(real T)`

  This is the saturated liquid density, which takes on temperature in K and returns liquid density in kg/m^3.

- `real wetst_cpl(real T)`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is the saturated liquid specific heat at constant pressure, which takes on temperature in K and returns liquid specific heat in J/kg/K.

- `real wetst_mul(real T)`

  This is the liquid dynamic viscosity, which takes on Temperature in K and returns dynamic viscosity in kg/m/s.

- `real wetst_ktl(real T)`

  This is the liquid thermal conductivity, which takes on temperature in K and returns thermal conductivity in W/m/K.

- `real wetst_surft(real T)`

  This is the liquid surface tension, which takes on Temperature in K and returns surface tension N/m.

At the end of the code you must define a structure of type `WS_Functions` whose members are pointers to the principle functions listed previously. The structure is of type `WS_Functions` and its name is `WetSteamFunctionList`.

```
UDF_EXPORT  WS_Functions  WetSteamFunctionList  =
 {
     wetst_init,     /*initialization  function*/
     wetst_satP,     /*Saturation  pressure*/
     wetst_satT,     /*Saturation  temperature*/
     wetst_eosP,     /*equation  of  state*/
     wetst_eosRHO,   /*equation  of  state*/
     wetst_hv,       /*vapor  enthalpy*/
     wetst_sv,       /*vapor  entropy*/
     wetst_cpv,      /*vapor  isobaric  specific  heat*/
     wetst_cvv,      /*vapor  isochoric  specific  heat*/
     wetst_muv,      /*vapor  dynamic  viscosity*/
     wetst_ktv,      /*vapor  thermal  conductivity*/
     wetst_rhol,     /*sat.  liquid  density*/
     wetst_cpl,      /*sat.  liquid  specific  heat*/
     wetst_mul,      /*sat.  liquid  viscosity*/
     wetst_ktl,      /*sat.  liquid  thermal  conductivity*/
     wetst_surft     /*liquid  surface  tension*/
 };
```

26.6.3. Compiling Your UDWSPF and Building a Shared Library File

This section presents the steps you will need to follow to compile your UDWSPF C code and build a shared library file. This process requires the use of a C compiler. Most UNIX operating systems provide a C compiler as a standard feature. If you are using a PC, you will need to ensure that a C++ compiler is installed before you can proceed (e.g., Microsoft Visual C++, v6.0 or higher).

Important

To use the UDWSPF you will need to first build the UDWSPF library by compiling your UDWSPF C code and then loading the library into the ANSYS FLUENT code.

The UDWSPF shared library is built in the same way that the ANSYS FLUENT executable itself is built. Internally, a script called `Makefile` is used to invoke the system C compiler to build an object code library that contains the native machine language translation of your higher-level C source code. This shared library is then loaded into ANSYS FLUENT (either at runtime or automatically when a case file is read) by a process called *dynamic loading*. The object libraries are specific to the computer architecture being used, as well as to the particular version of the ANSYS FLUENT executable being run. The libraries must, therefore, be rebuilt any time ANSYS

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUENT is upgraded, when the computer's operating system level changes, or when the job is run on a different type of computer.

The general procedure for compiling UDWSPF C code is as follows:

- Place the UDWSPF C code in your working directory (i.e., where your case file resides).
- Launch ANSYS FLUENT.
- Read your case file into ANSYS FLUENT.
- You can now compile your UDWSPF C code and build a shared library file using the commands provided in the text command interface (TUI):
  - Select the `define/models/multiphase/wet-steam` menu item.

    `define` → `models` → `multiphase` → `wet-steam`
  - Select the `compile-user-defined-wetsteam-functions` option.
  - Enter the compiled UDWSPF library name.

    The name given here is the name of the directory where the shared library (e.g., `libudf`) will reside. For example, if you hit `<Enter>` then a directory should exist with the name `libudf`, and this directory will contain library file called `libudf`. If, however, you type a new library name such as `mywetsteam`, then a directory called `mywetsteam` will be created and it will contain the library `libudf`.
  - Continue on with the procedure when prompted.
  - Enter the C source file names.

    Important

    Ideally you should place all of your functions into a single file. However, you can split them into separate files if desired.

  - Enter the header file names, if applicable. If you do not have an extra header file, then press `<Enter>` when prompted.

    ANSYS FLUENT will then start compiling the UDWSPF C code and put it in the appropriate architecture directory.

26.6.4. Loading the UDWSPF Shared Library File

To load the UDWSPF library, perform the following steps:

- Go to the `define/models/multiphase/wet-steam` menu item in the text user interface.

  `define` → `models` → `multiphase` → `wet-steam`
- Select the `load-unload-user-defined-wetsteam-library` option and follow the procedure when prompted.

  If the loading of the UDWSPF library is successful, you will see a message similar to the following:

```
Opening user-defined wet steam library "libudf"...
Library "libudf/lnx86/2d/libudf.so" opened

Setting material properties to Wet-Steam...
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        Initializing user defined material properties...
```

26.6.5. UDWSPF Example

This section describe a simple UDWSPF. You can use this example as a the basis for your own UDWSPF code. For approximate calculations at low pressure, the simple ideal-gas equation of state and constant isobaric specific heat is assumed and used. The properties at the saturated liquid line and the saturated vapor line used in this example are similar to the one used by ANSYS FLUENT.

```
/*********************************************************************/
 /* User Defined Wet Steam Properties:
   EOS    : Ideal Gas Eq.
   Vapor Sat. Line : W.C.Reynolds tables (1979)
   Liquid Sat. Line: E. Eckert & R. Drake book (1972)

   Use ideal-gas EOS with Steam properties
   to model wet steam condensation in low pressure nozzle
   Author: L. Zori
   Date : Jan. 29 2004
 */
 /*********************************************************************/
 #include "udf.h"
 #include "stdio.h"
 #include "ctype.h"
 #include "stdarg.h"

 /*Global Constants for this model*/
 real ws_TPP = 338.150 ;
 real ws_aaa = 0.01 ;
 real cpg = 1882.0;/* Cp-vapor at low-pressure region*/

 DEFINE_ON_DEMAND(I_do_nothing)
 {
    /* This is a dummy function to allow us to use */
    /* the Compiled UDFs utility      */
 }


 void wetst_init(Domain *domain)
 {
    /*
      You must initialize these material property constants..
      they will be used in the wet steam model in fluent
    */
    ws_Tc = 647.286  ;/*Critical Temp. */
    ws_Pc = 22089000.00 ;/*Critical Pressure */
    mw_f  = 18.016 ;/*fluid droplet molecular weight (water) */
    Rgas_v  = 461.50 ;/*vapor Gas Const*/
 }

 real wetst_satP(real T)
 {
    real psat;
    real SUM=0.0;
    real pratio;
    real F;
    real a1 = -7.41924200 ;
    real a2 = 2.97210000E-01;
    real a3 = -1.15528600E-01;
    real a4 = 8.68563500E-03;
    real a5 = 1.09409899E-03;
    real a6 = -4.39993000E-03;
    real a7 = 2.52065800E-03;
    real a8 = -5.21868400E-04;
    if (T > ws_Tc) T = ws_Tc ;
    F  = ws_aaa*(T - ws_TPP)  ;
    SUM = a1 + F*(a2+ F*(a3+ F*(a4+ F*(a5+ F*(a6+ F*(a7+ F*a8)))))) ;
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   pratio = (ws_Tc/T - 1.0)*SUM;
   psat  = ws_Pc *exp(pratio) ;
  return psat; /*Pa */
}


real
wetst_satT(real P, real T)
{
   real tsat;
   real dT, dTA,dTM,dP,p1,p2,dPdT;
   int i ;
     for (i=0; i25; ++i)
      {
        if (T > ws_Tc) T = ws_Tc-0.5;

        p1= wetst_satP(T)  ;
        p2= wetst_satP(T+0.1) ;
        dPdT = (p2-p1)/0.1 ;

        dP = P - p1 ;

        dT = dP/dPdT  ;

        dTA = fabs(dT);
        dTM = 0.1*T;
        if (dTA > dTM) dT=dT*dTM/dTA ;
        T = T + dT;
        if (fabs(dT) TEMP_eps*T) break;
        }
     tsat = T;
   return tsat; /*K */
}

real
wetst_eosP(real rho, real T)
{
   real P;

   P = rho* Rgas_v * T ;

  return P; /*Pa */
}


real
wetst_eosRHO(real P, real T)
{
    real rho;

    rho = P/(Rgas_v * T) ;

    return rho; /*kg/m3 */
}


real
wetst_cpv(real T, real rho)
{
    real cp;

    cp = cpg ;

    return cp; /* (J/Kg/K) */
}

real
wetst_cvv(real T, real rho)
{
   real cv;

   cv = wetst_cpv(T,rho) - Rgas_v ;
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   return cv; /* (J/Kg/K) */
}

real
wetst_hv(real T,real rho)
{
   real h;

   h = T* wetst_cpv(T,rho) ;

  return h; /* (J/Kg) */
}



real
wetst_sv(real T, real rho)
{
   real s ;

   real TDatum=288.15;
   real PDatum=1.01325e5;
   s=wetst_cpv(T,rho)*log(T/TDatum)+
   Rgas_v*log(PDatum/(Rgas_v*T*rho));

 return s; /* (J/Kg/K) */
}


real
wetst_muv(real T, real rho)
{
   real muv;

   muv=1.7894e-05 ;

   return muv; /* (Kg/m/s) */
}

real
wetst_ktv(real T, real rho)
{
   real ktv;

   ktv=0.0242 ;

   return ktv; /* W/m/K */
}


real
wetst_rhol(real T)
{
   real rhol;

   real SUM = 0.0;
   int ii;
   int i;
   real rhoc = 317.0;
   real D[8];

   D[0] =  3.6711257 ;
   D[1] = -2.8512396E+01 ;
   D[2] =  2.2265240E+02 ;
   D[3] = -8.8243852E+02 ;
   D[4] =  2.0002765E+03 ;
   D[5] = -2.6122557E+03 ;
   D[6] =  1.8297674E+03 ;
   D[7] = -5.3350520E+02 ;
   if (T > ws_Tc) T = ws_Tc ;
   for(ii=0;ii 8;++ii)
     {
        i = ii+1 ;
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        SUM += D[ii] * pow((1.0 - T/ws_Tc), i/3.0);
      }
   rhol = rhoc*(1.0+SUM);

   return rhol; /* (Kg/m3) */
}


real
wetst_cpl(real T)
{
   real cpl;

  real a1= -36571.6 ;
   real a2= 555.217 ;
   real a3= -2.96724 ;
   real a4= 0.00778551;
   real a5= -1.00561e-05;
   real a6= 5.14336E-09;

   if (T > ws_Tc) T = ws_Tc ;
   cpl = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*a6)))) ;

   return cpl; /* (J/Kg/K) */
}


real
wetst_mul(real T)
{
   real mul;

   real a1= 0.530784;
   real a2= -0.00729561;
   real a3= 4.16604E-05 ;
   real a4= -1.26258E-07;
   real a5= 2.13969E-10;
   real a6= -1.92145E-13;
   real a7= 7.14092E-17;

   if (T > ws_Tc) T = ws_Tc ;
   mul = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*(a6+ T*a7))))) ;

   return mul; /* (Kg/m/s) */
}

real
wetst_ktl(real T)
{
 real ktl;

   real a1= -1.17633;
   real a2= 0.00791645;  real a3= 1.48603E-05;
   real a4= -1.31689E-07;
   real a5= 2.47590E-10;
   real a6= -1.55638E-13;

   if (T > ws_Tc) T = ws_Tc ;
   ktl = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*a6)))) ;

 return ktl; /* W/m/K */
}

real
wetst_surft(real T)
{
   real sigma;

   real Tr;
   real a1= 82.27  ;
   real a2= 75.612;
   real a3= -256.889 ;
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    real a4= 95.928;

    if (T > ws_Tc) T = ws_Tc ;
    Tr  = T/ws_Tc ;
    sigma = 0.001*(a1 + Tr*(a2+ Tr*(a3+ Tr*a4))) ;

    return sigma ;/* N/m */
}

/* do not change the order of the function list */
UDF_EXPORT WS_Functions WetSteamFunctionList =
  {
  wetst_init,     /*initialization  function*/
  wetst_satP,     /*Saturation  pressure*/
  wetst_satT,     /*Saturation  temperature*/
  wetst_eosP,     /*equation  of  state*/
  wetst_eosRHO,   /*equation  of  state*/
  wetst_hv,       /*vapor  enthalpy*/
  wetst_sv,       /*vapor  entropy*/
  wetst_cpv,      /*vapor  isobaric  specific  heat*/
  wetst_cvv,      /*vapor  isochoric  specific  heat*/
  wetst_muv,      /*vapor  dynamic  viscosity*/
  wetst_ktv,      /*vapor  thermal  conductivity*/
  wetst_rhol,     /*sat.  liquid  density*/
  wetst_cpl,      /*sat.  liquid  specific  heat*/
  wetst_mul,      /*sat.  liquid  viscosity*/
  wetst_ktl,      /*sat.  liquid  thermal  conductivity*/
  wetst_surft     /*liquid  surface  tension*/
 };
/*********************************************************************/
```

26.7. Solution Strategies for Multiphase Modeling

For additional information, please see the following sections:

26.7.1. Coupled Solution for Multiphase Flows
26.7.2. Setting Initial Volume Fractions
26.7.3. VOF Model
26.7.4. Mixture Model
26.7.5. Eulerian Model
26.7.6. Wet Steam Model

26.7.1. Coupled Solution for Multiphase Flows

In multiphase flow, the phasic momentum equations, the shared pressure, and the phasic volume fraction equations are highly coupled. Traditionally, these equations have been solved in a segregated fashion using some variation of the SIMPLE algorithm to couple the shared pressure with the momentum equations. This is attained by effectively transforming the total continuity into a shared pressure. The ANSYS FLUENT **Phase Coupled SIMPLE** algorithm has been successfully implemented and solves a wide range of multiphase flows. However, coupling the linearized system of equations in an implicit manner would offer a more robust alternative to the segregated approach.

One of the fundamental problems is that the resulting matrix is not symmetric and that the continuity constraint may contribute to a zero diagonal block, making the solution difficult to obtain. One way to circumvent this problem is to use direct solvers, but these are too expensive for large industrial cases. In addition, we need to avoid a zero diagonal, resulting from the continuity constraint, and like the segregated solver, we need to construct a pressure correction equation. In multiphase, we also have the additional problem of the vanishing phase, which for the coupled solver is important to ensure some continuity in the coefficients. Like the **Phase Coupled**, we use a Rhie and Chow type of scheme to calculate volume fluxes and to provide proper coupling between velocity and pressure, thus avoiding unphysical oscillations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Consider a single-phase system and let us denote the velocity correction components in the three Cartesian directions by u', v', and w' with p' denoting the shared pressure correction. These are discrete variables and can be expressed in the form p', U'. The linear system that is generated by the single-phase coupled solver is of the form

$$\begin{pmatrix} A_p & C_U \\ B_U & A_U \end{pmatrix} \begin{pmatrix} p' \\ U' \end{pmatrix} = \begin{pmatrix} S_p \\ S_U \end{pmatrix} \tag{26–10}$$

For a notation in component form (p', u', v', w')

$$\begin{pmatrix} A_{pp} & C_u & C_v & C_w \\ B_u & A_{uu} & A_{uv} & A_{uw} \\ B_v & A_{vu} & A_{vv} & A_{vw} \\ B_w & A_{wu} & A_{wv} & A_{ww} \end{pmatrix} \begin{pmatrix} p' \\ u' \\ v' \\ w' \end{pmatrix} = \begin{pmatrix} S_p \\ S_u \\ S_v \\ S_w \end{pmatrix} \tag{26–11}$$

Now let us consider a multiphase system of n-phases and denote the phasic velocity correction components in the three Cartesian directions by u'_k, v'_k, and w'_k where the subscript k represents the phase notation, p' denotes the shared pressure correction and α'_k denotes the volume fraction correction (ANSYS FLUENT can solve in both correction form for velocity and volume fraction and noncorrection form). For simplicity the matrix will be shown for two phases. The vector solution is of the form $(p', u'_1, v'_1, w'_1, u'_2, v'_2, w'_2, \alpha'_2)$ or in a shorter notation $(p', U'_1, U'_2, \alpha'_2)$. The linear system would be an extension of the one generated by the coupled solver shown by *Equation 26–10* (p. 1188).

$$\begin{pmatrix} A_p & C_{U1} & C_{U2} & D_{\alpha 2} \\ B_{U1} & A_{U1} & A_{U12} & D_{U1} \\ B_{U2} & A_{U21} & A_{U2} & D_{U2} \\ E_p & E_{U1} & E_{U2} & A_{\alpha 2} \end{pmatrix} \begin{pmatrix} p' \\ U'_1 \\ U'_2 \\ \alpha'_2 \end{pmatrix} = \begin{pmatrix} S_p \\ S_{U1} \\ S_{U2} \\ S_{\alpha 2} \end{pmatrix} \tag{26–12}$$

This system can be easily generalized to n phases. The components of this matrix are also matrices.

For large problems we need to resort to iterative solvers. The ANSYS FLUENT AMG Coupled solver with an ILU smoother has proved to be a robust method. Most coupled solvers also need a pseudo stepping method, adding more diagonal dominance to the matrix. Our method here is to use under-relaxation factors for momentum, which is equivalent to time stepping in steady flows. Similar to that of the single phase, we have introduced a steady Courant Number instead of an under-relaxation for velocities. Having this control is important when using second order numerical schemes in the convective terms.

For the sake of simplicity, input parameters for the **Multiphase Coupled** solver are similar to the single-phase solver. We have the options for solving the whole system including volume fraction, or to treat the volume fraction solution in a segregated manner while preserving the pressure-velocity coupling for all phases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Multiphase is more implicit than single phase and generally may need more under-relaxation, hence using the same values as single phase may not be ideal.

See *Selecting the Solution Method* (p. 1193) for information about applying the various algorithms.

26.7.2. Setting Initial Volume Fractions

Once you have initialized the flow (as described in *Initializing the Solution* (p. 1239)), you can define the initial distribution of the phases. For a transient simulation, this distribution will serve as the initial condition at $t=0$; for a steady-state simulation, setting an initial distribution can provide added stability in the early stages of the calculation.

You can patch an initial volume fraction for each secondary phase using the *Patch Dialog Box* (p. 1939).

Solution Initialization → **Patch...**

If the region in which you want to patch the volume fraction is defined as a separate cell zone, you can simply patch the value there. Otherwise, you can create a cell "register" that contains the appropriate cells and patch the value in the register. See *Patching Values in Selected Cells* (p. 1242) for details.

Solution strategies for the VOF, mixture, and Eulerian models are provided in *VOF Model* (p. 1189), *Mixture Model* (p. 1192), and *Eulerian Model* (p. 1192), respectively.

26.7.3. VOF Model

Several recommendations for improving the accuracy and convergence of the VOF solution are presented here.

26.7.3.1. Setting the Reference Pressure Location

The site of the reference pressure can be moved to a location that will result in less round-off in the pressure calculation. By default, the reference pressure location is the center of the cell at or closest to the point (0,0,0). You can move this location by specifying a new **Reference Pressure Location** in the *Operating Conditions Dialog Box* (p. 1806).

Cell Zone Conditions → **Operating Conditions...**

The position that you choose should be in a region that will always contain the least dense of the fluids (e.g., the gas phase, if you have a gas phase and one or more liquid phases). This is because variations in the static pressure are larger in a more dense fluid than in a less dense fluid, given the same velocity distribution. If the zero of the relative pressure field is in a region where the pressure variations are small, less round-off will occur than if the variations occur in a field of large nonzero values. Thus in systems containing air and water, for example, it is important that the reference pressure location be in the portion of the domain filled with air rather than that filled with water.

26.7.3.2. Pressure Interpolation Scheme

For all VOF calculations, you should use the body-force-weighted pressure interpolation scheme or the PRESTO! scheme.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Controls

26.7.3.3. Discretization Scheme Selection for the Implicit and Explicit Formulations

When the implicit scheme is used, the available options for **Volume Fraction Discretization** are

Solution Methods

- **First Order Upwind**
- **Second Order upwind**
- **Compressive** (available exclusively when **Zonal Discretization** is enabled in the **Multiphase Model** dialog box)
- **Modified HRIC**
- **BGM** (steady state only)
- **QUICK**

When the explicit scheme is used, the available options for **Volume Fraction Discretization** are

- **Geo-Reconstruct**
- **CICSAM**
- **Compressive** (available exclusively when **Zonal Discretization** is enabled in the **Multiphase Model** dialog box)
- **Modified HRIC**
- **QUICK**

When using the explicit scheme, **First Order Upwind**, **Second Order upwind**, and **Donor-Acceptor** can be made available under **Volume Fraction Discretization** by using the following text command:

`solve` → `set` → `expert`

You will be asked a series of questions, one of which is

```
Allow selection of all applicable discretization schemes? [no]
```

to which you will respond `yes`.

Important

You are encouraged to use the CICSAM scheme, as it gives a sharper interface than the modified HRIC scheme.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

In VOF modeling, using a high-order discretization scheme for the momentum transport equations may reduce the stability of the solution compared to cases using first-order discretization. In such situations, it is recommended to use a low-order variant of Rhie-Chow face flux interpolation, which can be turned on using the text command:

`solve` → `set` → `numerics`

When asked to `disable high order Rhie-Chow flux?[no]`, enter `yes`.

When using the **Compressive** spatial discretization scheme, it is recommended that you use step-wise sharpening after the flow transitions from the diffused zone to the sharpening zone. In other words, you would want to transition from first order to second order, followed by a transition from second order to compressive. You would want to take this approach if the flow has a smooth transition from a sharp interfacial zone to a diffused interfacial zone. However, if the flow has a nonuniform transition from a diffused interfacial zone to a sharp interfacial zone, this might create unphysical sharpening of the interface if not handled properly, especially for transient cases. For example, transitioning from a slope limiter of 2 (compressive) to a slope limiter of 0 or 1 (first order or second order) might be acceptable, but a first order to compressive transition might create unphysical sharpening of the interface.

Important

The **BGM** scheme produces a sharp interface, which may result in poor convergence in some cases. In such situations, we recommend you use a low value for the VOF under-relaxation. In addition, you can start with the **Compressive** or **Modified HRIC** scheme and then switch to the **BGM** scheme.

26.7.3.4. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations

Another change that you should make to the solver settings is in the pressure-velocity coupling scheme and under-relaxation factors that you use. The PISO scheme is recommended for transient calculations in general. Using PISO allows for increased values on all under-relaxation factors, without a loss of solution stability. You can generally increase the under-relaxation factors for all variables to 1 and expect stability and a rapid rate of convergence (in the form of few iterations required per time step). For calculations on tetrahedral or triangular meshes, an under-relaxation factor of 0.7–0.8 for pressure is recommended for improved stability with the PISO scheme.

Solution Controls

As with any ANSYS FLUENT simulation, the under-relaxation factors will need to be decreased if the solution exhibits unstable, divergent behavior with the under-relaxation factors set to 1. Reducing the time step is another way to improve the stability.

26.7.3.5. Under-Relaxation for the Steady-State Formulation

If you are using the steady-state implicit VOF scheme, the under-relaxation factors for all variables should be set to values between 0.2 and 0.5 for improved stability.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.7.4. Mixture Model

26.7.4.1. Setting the Under-Relaxation Factor for the Slip Velocity

You should begin the mixture calculation with a low under-relaxation factor for the slip velocity. A value of 0.2 or less is recommended. If the solution shows good convergence behavior, you can increase this value gradually.

26.7.4.2. Calculating an Initial Solution

For some cases (e.g., cyclone separation), you may be able to obtain a solution more quickly if you compute an initial solution without solving the volume fraction and slip velocity equations. Once you have set up the mixture model, you can temporarily disable these equations and compute an initial solution.

Solution Controls

In the *Equations Dialog Box* (p. 1904), deselect **Volume Fraction** and **Slip Velocity** in the **Equations** list. You can then compute the initial flow field. Once a converged flow field is obtained, turn the **Volume Fraction** and **Slip Velocity** equations back on again, and compute the mixture solution.

26.7.5. Eulerian Model

26.7.5.1. Calculating an Initial Solution

To improve convergence behavior, you may want to compute an initial solution before solving the complete Eulerian multiphase model. There are three methods you can use to obtain an initial solution for an Eulerian multiphase calculation:

- Set up and solve the problem using the mixture model (with slip velocities) instead of the Eulerian model. You can then enable the Eulerian model, complete the setup, and continue the calculation using the mixture-model solution as a starting point.
- Set up the Eulerian multiphase calculation as usual, but compute the flow for only the primary phase. To do this, deselect **Volume Fraction** in the **Equations** list in the *Equations Dialog Box* (p. 1904). Once you have obtained an initial solution for the primary phase, turn the volume fraction equations back on and continue the calculation for all phases.
- Use the mass flow inlet boundary condition to initialize the flow conditions. It is recommended that you set the value of the volume fraction close to the value of the volume fraction at the inlet.
- At the beginning of the solution, a lower time step is recommended to obtain convergence.
- If using the volume fraction explicit scheme, do not start with a large Courant number at the beginning of your run.
- If using the volume fraction explicit scheme, start a run with a lower time step and then increase the time step size. Alternatively, this could be done by using variable time stepping, which would increase the time step size based on the input parameters.
- For problems involving a free surface or sharp interfaces between the phases, it is recommended that you use the **symmetric** drag law, available in the *Phase Interaction Dialog Box* (p. 1791).
- Variable time stepping is not recommended for compressible flows.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

You should *not* try to use a single-phase solution obtained without the mixture or Eulerian model as a starting point for an Eulerian multiphase calculation. Doing so will not improve convergence, and may make it even more difficult for the flow to converge.

26.7.5.2. Temporarily Ignoring Lift and Virtual Mass Forces

If you are planning to include the effects of lift and/or virtual mass forces in a steady-state Eulerian multiphase simulation, you can often reduce stability problems that sometimes occur in the early stages of the calculation by temporarily ignoring the action of the lift and the virtual mass forces. Once the solution without these forces starts to converge, you can interrupt the calculation, define these forces appropriately, and continue the calculation.

26.7.5.3. Selecting the Solution Method

There are three options to solve the coupled system of equations arising in multiphase flows. These options that are available in the **Solution Methods** task page are:

- **Phase Coupled SIMPLE**
- **Multiphase Coupled**
- **Full Multiphase Coupled**

The **Phase Coupled SIMPLE** (PC-SIMPLE) is an extension of the SIMPLE algorithm [*61*] (p. 2212) to multiphase flows. The velocities are solved coupled by phases in a segregated fashion. Fluxes are reconstructed at the faces of the control volume and then a pressure correction equation is built based on total continuity. The coefficients of the pressure correction equations come from the coupled per phase momentum equations. This method has proven to be robust and it is the only method available for all previous versions of ANSYS FLUENT.

The **Multiphase Coupled** solves all equations for phase velocity corrections and shared pressure correction simultaneously [*25*] (p. 2210). These methods incorporate the lift forces and the mass transfer terms implicitly into the general matrix. This method works very efficiently in steady state situations, or for transient problems when larger time steps are required.

The **Full Multiphase Coupled** option couples velocity corrections, shared pressure corrections, and the correction for volume fraction simultaneously. Theoretically it should be more efficient, however it may have some drawbacks in robustness and CPU time usage. The robustness issue stems from the lack of control of the solution of the volume fraction equation. The continuity constraint (sum of all volume fractions equals 1, and individual values limited between zero and one) cannot be enforced exactly during inner solver iterations, and slight variations from the physical limits may lead to divergence. Research is undergoing in this area to improve the method. The method works well in dilute situations.

26.7.5.4. Discretization Scheme Selection for the Implicit and Explicit Formulations

When the implicit scheme is used, the available options for **Volume Fraction Discretization** are

- **First Order Upwind**
- **Compressive** (available exclusively when the **Immiscible Fluid Model** and **Zonal Discretization** is enabled in the **Multiphase Model** dialog box)
- **QUICK**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the explicit scheme is used, the available options for **Volume Fraction Discretization** are

- **First Order Upwind**
- **Geo-Reconstruct**
- **CICSAM**
- **Compressive** (available exclusively when the **Immiscible Fluid Model** and **Zonal Discretization** is enabled in the **Multiphase Model** dialog box)
- **Modified HRIC**
- **QUICK**

When using the explicit scheme **Second Order upwind**, and **Donor-Acceptor** can be made available under **Volume Fraction Discretization** by using the following text command:

`solve` → `set` → `expert`

You will be asked a series of questions, one of which is

```
Allow selection of all applicable discretization schemes? [no]
```

to which you will respond `yes`.

26.7.5.5. Using W-Cycle Multigrid

For problems involving a packed-bed granular phase with very small particle sizes (on the order of 10 μm), convergence can be obtained by using the W-cycle multigrid for the pressure. In the **Multigrid** tab, under **Fixed Cycle Parameters** in the *Advanced Solution Controls Dialog Box* (p. 1906), you may need to use higher values for **Pre-Sweeps**, **Post-Sweeps**, and **Max Cycles**. When you are choosing the values for these parameters, you should also increase the **Verbosity** to `1` in order to monitor the AMG performance; i.e., to make sure that the pressure equation is solved to a desired level of convergence within the AMG solver during each global iteration. See *Defining the Phases for the Eulerian Model* (p. 1154) for more information about granular phases, and The V and W Cycles in the Theory Guide and *Modifying Algebraic Multigrid Parameters* (p. 1319) for details about multigrid cycles.

26.7.5.6. Including the Anisotropic Drag Law

When using the anisotropic drag law (*Including the Immiscible Fluid Model* (p. 1176)), it is recommended that you start the solution with a lower anisotropy ratio. After you let your solution run for some time, you can then increase the ratio by reducing the friction factor in the tangential direction. Note that You can also start the solution with the symmetric drag law, then change to the anisotropic drag law.

Using a smaller under-relaxation for pressure and momentum may also help in convergence for cases with a higher anisotropy ratio.

If the flow for a particular phase is important in both directions (normal and tangential to the interface), use a lower anisotropy ratio, between 100-1000. A higher anisotropy ratio might cause an unstable solution for such cases. For a higher anisotropy ratio of more than 1000, a smaller under-relaxation for pressure and momentum is recommended. When using the coupled multiphase solver, if the solution is unstable with a higher anisotropy ratio, then reducing the courant number may be beneficial. `Anisotropic Drag Method` [1], with `Viscosity option` [2] is recommended for a higher viscosity ratio.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.7.6. Wet Steam Model

26.7.6.1. Boundary Conditions, Initialization, and Patching

When you use the wet steam model (described in Wet Steam Model Theory in the Theory Guide, and *Setting Up the Wet Steam Model* (p. 1178)), the following two field variables will show up in the inflow, outflow boundary dialog boxes, and in the **Solution Initialization** task page and **Patch** dialog boxes.

- **Liquid Mass Fraction** (or the wetness factor)

  In general, for dry steam entering flow boundaries the wetness factor is zero.

- **Log10 (Droplets Per Unit Volume)**

  In general this value is set to zero, indicating zero droplets entering the domain.

26.7.6.2. Solution Limits for the Wet Steam Model

When you activate the wet steam model for the first time, a message is displayed indicating that the **Minimum Static Temperature** should be adjusted to 273 K since the accuracy of the built-in steam data is not guaranteed below a value of 273 K. If you use your own steam property functions, you can adjust this limit to whatever is permissible for your data.

To adjust the temperature limits, go to the **Solution Limits** dialog box.

Solution Controls → **Limits...**

The default maximum wetness factor or liquid mass fraction (β) is set to 0.1. In general, during the convergence process, it is common that this limit will be reached, but eventually the wetness factor will drop below the value of 0.1. However, in cases where the limit must be adjusted, you can do so using the text user interface.

`define` → `models` → `multiphase` → `wet-steam` → `set` → `max-liquid-mass-fraction`

Important

Note that the maximum wetness factor should not be set beyond 0.2 since the present model assumes a low wetness factor. When the wetness factor is greater than 0.1, the solution tends to be less stable due to the large source terms in the transport equations. Thus, the maximum wetness factor has been set to a default value of 0.1, which corresponds to the fact that most nozzle and turbine flows will have a wetness factor less than 0.1.

26.7.6.3. Solution Strategies for the Wet Steam Model

If you face convergence difficulties while solving wet steam flow, try to initially lower the CFL value and use first-order discretization schemes for the solution. If you are still unable to obtain a converged solution, then try the following solver settings:

1. Lower the under-relaxation factor for the wet steam equation below the current set value. The under-relaxation factor can be found in the **Solution Controls** task page.

   Solution Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Solve for an initial solution with no condensation. Once you have obtained a proper initial solution, turn on the condensation.

   To turn condensation on or off, go to the **Solution Controls** task page.

   Solution Controls

   In the **Equations** dialog box, deselect **Wet Steam** in the **Equations** list. When doing so, you are preventing condensation from taking place while still computing the flow based on steam properties. Once a converged flow field is obtained, turn the **Wet Steam** equation back on again and compute the mixture solution.

26.8. Postprocessing for Multiphase Modeling

Each of the three general multiphase models provides a number of additional field functions that you can plot or report. You can also report flow rates for individual phases for all three models, and display velocity vectors for the individual phases in a mixture or Eulerian calculation.

Information about these postprocessing topics is provided in the following subsections:

26.8.1. Model-Specific Variables
26.8.2. Displaying Velocity Vectors
26.8.3. Reporting Fluxes
26.8.4. Reporting Forces on Walls
26.8.5. Reporting Flow Rates

26.8.1. Model-Specific Variables

When you use one of the general multiphase models, some additional field functions will be available for postprocessing, as listed in this section. Most field functions that are available in single phase calculations will be available for either the mixture or each individual phase, as appropriate for the general multiphase model and specific options that you are using. See *Field Function Definitions* (p. 1529) for a complete list of field functions and their definitions. *Displaying Graphics* (p. 1379) and *Reporting Alphanumeric Data* (p. 1509) explain how to generate graphics displays and reports of data.

26.8.1.1. VOF Model

For VOF calculations you can generate graphical plots or alphanumeric reports of the following additional item:

- **Volume fraction** (in the **Phases...** category)

  This item is available for each phase.

The variables that are not phase specific are available (e.g., variables in the **Pressure...** and **Velocity...** categories) represent mixture quantities. Thermal quantities will be available only for calculations that include the energy equation.

26.8.1.2. Mixture Model

For calculations with the mixture model, you can generate graphical plots or alphanumeric reports of the following additional items:

- **Diameter** (in the **Properties...** category)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This item is available only for secondary phases.

- **Volume fraction** (in the **Phases...** category)

  This item is available only for secondary phases.

- **Interfacial Area Concentration** (in the **Interfacial Area Concentration...** category)

  This item is available only for secondary phases.

The variables that are not phase specific are available (e.g., variables in the **Pressure...** category) represent mixture quantities. Thermal quantities will be available only for calculations that include the energy equation.

26.8.1.3. Eulerian Model

For Eulerian multiphase calculations you can generate graphical plots or alphanumeric reports of the following additional items:

- **Diameter** (in the **Properties...** category)

  This item is available only for secondary phases.

- **Granular Conductivity** (in the **Properties...** category)

  This item is available only for granular phases.

- **Granular Pressure** (in the **Granular Pressure...** category)

  This item is available only for granular phases.

- **Granular Temperature** (in the **Granular Temperature...** category)

  This item is available only for granular phases.

- **Volume fraction** (in the **Phases...** category)

  This item is available only for secondary phases.

- **Interfacial Area Concentration** (in the **Interfacial Area Concentration...** category)

  This item is available only for secondary phases.

The availability of turbulence quantities will depend on which multiphase turbulence model you used in the calculation. Thermal quantities will be available (on a per-phase basis) only for calculations that include the energy equation.

More advanced options for the mixture phase are available under the **Phases...** category, allowing you to select from a list of variables to postprocess. To access the entire list in the GUI, type the following text command:

`solve` → `set` → `expert`

Retain most of the default settings, except when asked to `Keep temporary solver memory from being freed?`. Answering `yes` to this question will expose a list under the **Phases** category for the mixture phase, one of which will be the **Phase ID**. Selecting this option allows you to plot contours of phase IDs for the volume fraction, which will facilitate phase distribution display when more than two phases are present for free surface calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

This option is available for all the multiphase models. However, note that only cell values should be plotted for this option. Make sure that the **Node Values** option is not selected as it will show the wrong phase ID contours at the interface.

26.8.1.4. Multiphase Species Transport

For calculations using species transport with either of the multiphase models, you can generate graphical plots or alphanumeric reports of the following additional items:

- **Mass Fraction of species-n** (in the **Species...** category)

  This item is available for each species.
- **Mole Fraction of species-n** (in the **Species...** category)

  This item is available for each species.
- **Molar Concentration of species-n** (in the **Species...** category)

  This item is available for each species.
- **Lam. Diff Coeff of species-n** (in the **Species...** category)

  This item is available for each species.
- **Eff. Diff. Coeff. of species-n** (in the **Species...** category)

  This item is available for each species.
- **Enthalpy of species-n** (in the **Species...** category)

  This item is available for each species.
- **Relative Humidity** (in the **Species...** category).
- **Turbulent Rate of Reaction-n** (in the **Reactions...** category)

  This item is available for each species.
- **Rate of Reaction** (in the **Reactions...** category).
- **Mass Transfer Rate n** (in the **Phase Interaction...** category)

  This item is available for each mass transfer mechanism that you defined.

Thermal quantities will be available only for calculations that include the energy equation.

26.8.1.5. Wet Steam Model

ANSYS FLUENT provides a wide range of postprocessing information related to the wet steam model.

The wet steam related items can be found in **Wet Steam....** category of the variable selection drop-down list that appears in the postprocessing dialog boxes.

- **Liquid Mass Fraction**
- **Liquid Mass Generation Rate**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Log10 (Droplets Per Unit Volume)**
- **Log10 (Droplets Nucleation Rate)**
- **Steam Density (Gas-Phase)**
- **Liquid Density (Liquid-Phase)**
- **Mixture Density**
- **Saturation Ratio**
- **Saturation Pressure**
- **Saturation Temperature**
- **Subcooled Vapor Temperature**
- **Droplet Surface Tension**
- **Droplet Critical Radius (microns)**
- **Droplet Average Radius (microns)**
- **Droplet Growth Rate (microns/s)**

26.8.1.6. Dense Discrete Phase Model

For postprocessing, both the DPM and the Eulerian multiphase capabilities are retained. In addition to usual DPM postprocessing (*Postprocessing for the Discrete Phase* (p. 1067)), you can display, for example, vector plots of the particle's velocity field. Make sure to select the discrete phase from the **Phase** drop-down list. For transient simulations which include the dense discrete phase model, you can display the following when the **Unsteady Statistics...** category is selected:

- **Mean Velocity**
- **Mean Volume Fraction**
- **Mean Phase Diameter**
- **RMS Velocity**
- **RMS Volume Fraction**
- **RMS Phase Diameter**

Important

For the **Unsteady Statistics...** category to appear in the postprocessing dialog boxes, make sure that **Data Sampling for Time Statistics** is enabled in the **Run Calculation** task page, and that you have performed the calculation.

26.8.2. Displaying Velocity Vectors

For mixture and Eulerian calculations, it is possible to display velocity vectors for the individual phases using the **Vectors** dialog box.

Graphics and Animations → **Vectors** → **Set Up...**

To display the velocity of a particular phase, select **Velocity** in the **Vectors of** drop-down list, and then select the desired phase in the **Phase** drop-down list. You can also choose **Relative Velocity** to display the phase velocity relative to a moving reference frame. To display the mixture velocity $\vec{v}_m$ (relevant for mixture

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model calculations only), select **Velocity** (or **Relative Velocity** for the mixture velocity relative to a moving reference frame), and **mixture** as the **Phase**. Note that you can color vectors by values of any available variable, for any phase you defined. To do so, make the appropriate selections in the **Color by** and following **Phase** drop-down lists.

26.8.3. Reporting Fluxes

When you use the **Flux Reports** dialog box to compute fluxes through boundaries, you will be able to specify whether the report is for the mixture or for an individual phase.

Reports → **Fluxes** → **Set Up...**

Select **mixture** in the **Phase** drop-down list at the bottom of the dialog box to report fluxes for the mixture, or select the name of a phase to report fluxes just for that phase.

26.8.4. Reporting Forces on Walls

For Eulerian calculations, when you use the **Force Reports** dialog box to compute forces or moments on wall boundaries, you will be able to specify the individual phase for which you want to compute the forces.

Reports → **Forces** → **Set Up...**

Select the name of the desired phase in the **Phase** drop-down list on the left side of the dialog box.

26.8.5. Reporting Flow Rates

You can obtain a report of mass flow rate for each phase (and the mixture) through each flow boundary using the `report/mass-flow` text command:

`report` → `mass-flow`

When you specify the phase of interest (the mixture or an individual phase), ANSYS FLUENT will list each zone, followed by the mass flow rate through that zone for the specified phase. An example is shown below.

```
/report> mf
(mixture water air)
domain id/name [mixture] air
zone 10 (spiral-press-outlet): -1.2330244
zone 3 (pressure-outlet): -9.7560663
zone 11 (spiral-vel-inlet): 0.6150589
zone 8 (spiral-wall): 0
zone 1 (walls): 0
zone 4 (velocity-inlet): 4.9132133

net mass-flow: -5.4608185
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 27: Modeling Solidification and Melting

This chapter describes how you can model solidification and melting in ANSYS FLUENT. For information about the theory behind the model, see "Solidification and Melting" in the Theory Guide. Information about using the model is organized into the following sections:

27.1. Setup Procedure
27.2. Procedures for Modeling Continuous Casting
27.3. Solution Procedure
27.4. Postprocessing

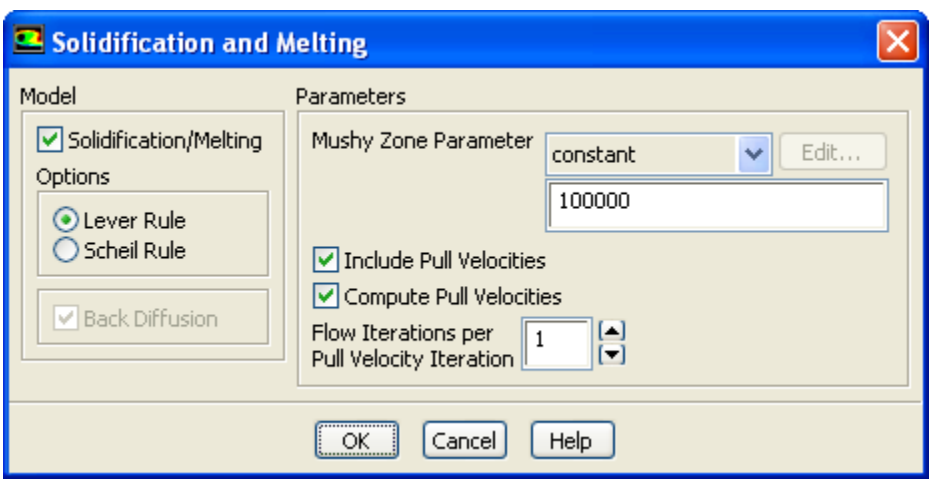
27.1. Setup Procedure

The procedure for setting up a solidification/melting problem is described below. (Note that this procedure includes only those steps necessary for the solidification/melting model itself; you will need to set up other models, boundary conditions, etc. as usual.)

1. To activate the solidification/melting model, enable the **Solidification/Melting** option in the **Solidification and Melting** dialog box (*Figure 27.1* (p. 1201)).

   Models → **Solidification & Melting** → **Edit...**

   Figure 27.1 The Solidification and Melting Dialog Box



   ANSYS FLUENT will automatically enable the energy equation, so you do not have to visit the **Energy** dialog box before turning on the solidification/melting model.

2. Under **Parameters**, specify the value of the **Mushy Zone Parameter** (A_{mush} in Equation 18–6) as a **constant**, or as a **user-defined** function. Please refer to DEFINE_SOLIDIFICATION_PARAMS in the UDF Manual for detailed information about the user-defined function.

   Values between 10^4 and 10^7 are recommended for most computations. The higher the value of the **Mushy Zone Parameter**, the steeper the damping curve becomes, and the faster the velocity drops

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to zero as the material solidifies. Very large values may cause the solution to oscillate as control volumes alternately solidify and melt with minor perturbations in liquid volume fraction.

3. If you want to include the pull velocity in your simulation (as described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide), enable the **Include Pull Velocities** option under **Parameters**.

4. If you are including pull velocities and you want ANSYS FLUENT to compute them (using Equation 18–22) based on the specified velocity boundary conditions, as described in Pull Velocity for Continuous Casting in the Theory Guide, enable the **Compute Pull Velocities** option and specify the number of **Flow Iterations Per Pull Velocity Iteration**.

Important

It is not necessary to have ANSYS FLUENT compute the pull velocities. See *Procedures for Modeling Continuous Casting* (p. 1204) for information about other approaches.

The default value of 1 for the **Flow Iterations Per Pull Velocity Iteration** indicates that the pull velocity equations will be solved after each iteration of the solver. If you increase this value, the pull velocity equations will be solved less frequently. You may want to increase the number of **Flow Iterations Per Pull Velocity Iteration** if the liquid fraction equation is almost converged (i.e., the position of the liquid-solid interface is not changing very much). This will speed up the calculation, although the residuals may jump when the pull velocities are updated.

5. Under **Options**, select either **Lever Rule** or **Scheil Rule**. See Species Equations in the Theory Guide for details.

Important

The **Lever Rule** and **Scheil Rule** options are available only when **Species Transport** is enabled in the **Species Model** dialog box.

---

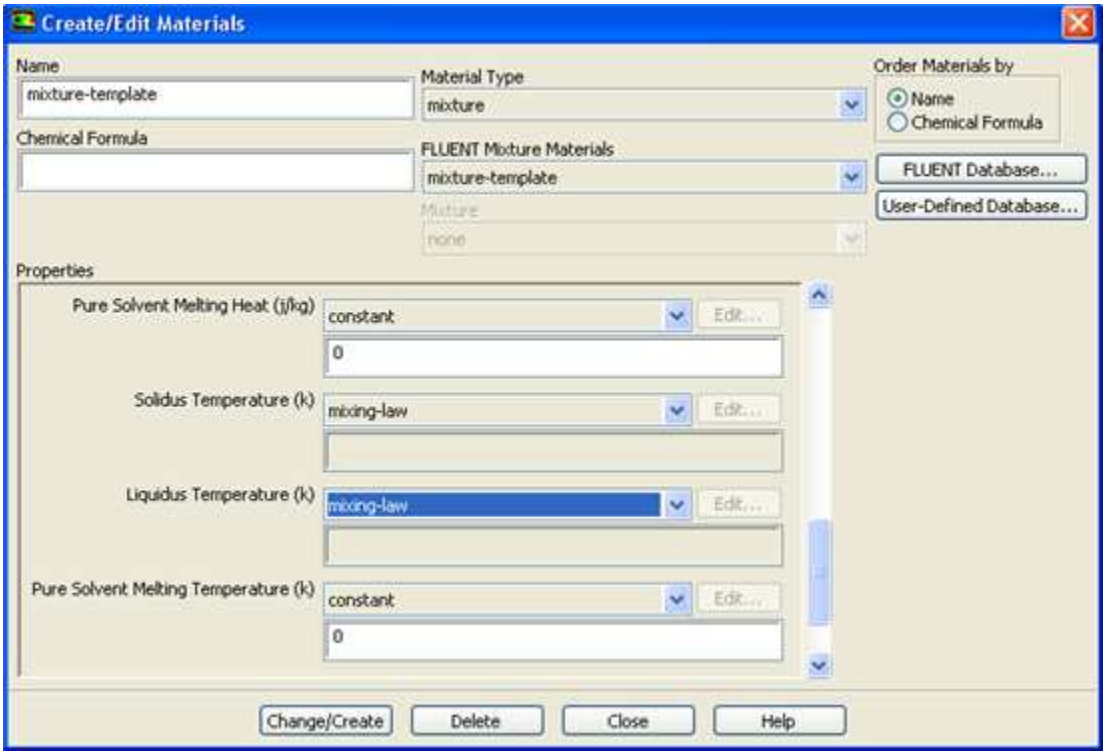
6. If you select **Scheil Rule**, then you can enable **Back Diffusion**. Enter either a **constant** or a **user-defined** function to specify the value of the **Back Diffusion Parameter** (γ in Equation 18–19). Please refer to DEFINE_SOLIDIFICATION_PARAMS in the UDF Manual for detailed information about the user-defined function. Note that the value for the **Back Diffusion Parameter** must be between 0 and 1.

7. In the *Create/Edit Materials Dialog Box* (p. 1736) (*Figure 27.2* (p. 1203)), specify the **Pure Solvent Melting Heat** (L in Equation 18–2), **Solidus Temperature** ($T_{solidus}$ in Equation 18–2), and **Liquidus Temperature** ($T_{liquidus}$ in Equation 18–2) for the material being used in your model.

Materials

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 27.2 The Create/Edit Materials Dialog Box for Melting and Solidification



If you are solving for species transport, you need to specify properties for the mixture, including the method by which the **Solidus Temperature** (T_{solidus} in Equation 18–8) and the **Liquidus Temperature** (T_{liquidus} in Equation 18–9) are calculated. The default method is the **mixing-law**, in which the solidus temperatures and the liquidus temperatures are calculated from the parameters provided for each solute (such as the slope of the liquidus line or partition coefficient). However, a **user-defined** function of type `DEFINE_PROPERTY` can be used to specify both of these temperatures. See the UDF Manual for examples of `DEFINE_PROPERTY`.

Important

It is highly recommended that you use the same method for specifying the **Solidus Temperature** and the **Liquidus Temperature**.

When defining the mixture, you will also specify the **Mass Diffusivity** ($D_{i,m,\text{liq}}$ in Equation 18–15 and Equation 18–18) and the **Eutectic Temperature** (T_{Eut} in Equation 18–10), as well as the **Pure Solvent Melting Heat** (L in Equation 18–4) and the **Pure Solvent Melting Temperature** (T_{melt} in Equation 18–8 and Equation 18–9). Note that the solvent is the last species listed under **Selected Species** in the **Species** dialog box.

For each solute, you have to specify the **Slope of Liquidus Line** (m_i in Equation 18–8 and Equation 18–9 in the Theory Guide) with respect to the concentration of the solute, the **Partition Coefficient** (K_i in Equation 18–8), the **Eutectic Mass Fraction** ($Y_{i,\text{Eut}}$ in Equation 18–10), and, if **Lever Rule** is selected in the **Solidification and Melting** dialog box, the coefficient for **Diffusion in Solid** ($D_{i,m,\text{sol}}$ in Equation 18–15). It is not necessary to specify m_i, K_i, $Y_{i,\text{Eut}}$, and $D_{i,m,\text{sol}}$ for the solvent.

8. Set the boundary conditions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Conditions...

In addition to the usual boundary conditions, consider the following:

- If you want to account for the presence of an air gap between a wall and an adjacent solidified region (as described in Contact Resistance at Walls in the Theory Guide), specify a nonzero value, a profile, or a user-defined function for **Contact Resistance** (R_c in Equation 18–23) under **Thermal Conditions** in the *Wall Dialog Box* (p. 1865).
- If you want to specify the gradient of the surface tension with respect to the temperature at a wall boundary, you can use the **Marangoni Stress** option for the wall **Shear Condition**. See *Marangoni Stress* (p. 305) for details.
- If you want ANSYS FLUENT to compute the pull velocities during the calculation, note how your specified velocity conditions are used in this calculation (see Pull Velocity for Continuous Casting the Theory Guide).

Procedures for Modeling Continuous Casting (p. 1204) contains additional information about modeling continuous casting. See *Solution Procedure* (p. 1205) and *Postprocessing* (p. 1205) for information about solving a solidification/melting model and postprocessing the results.

27.2. Procedures for Modeling Continuous Casting

As described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide, you can include the pull velocities in your solidification/melting calculation to model continuous casting. There are three approaches to modeling continuous casting in ANSYS FLUENT:

- Specify constant or variable pull velocities.

  To use this approach (the default), do not enable the **Compute Pull Velocities** option.

  If you use this approach, you will need to patch constant values or custom field functions for the pull velocities, after you initialize the solution.

  Solution Initialization → **Patch...**

  See *Patching Values in Selected Cells* (p. 1242) for details about patching values. Note that it is acceptable to patch values for the pull velocities in the entire domain, because the patched values will be used only if the liquid fraction, β, is less than 1.

- Have ANSYS FLUENT compute the pull velocities (using Equation 18–22) during the calculation, based on the specified velocity boundary conditions.

  To use this approach, enable the **Compute Pull Velocities** option. This method is computationally expensive, and is recommended only if the pull velocities are strongly dependent on the location of the liquid-solid interface.

  If you have ANSYS FLUENT compute the pull velocities, then there are no additional inputs or setup procedures beyond those presented in *Setup Procedure* (p. 1201).

- Have ANSYS FLUENT compute the pull velocities just once, and then use those values for the remainder of the calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use this approach, perform one iteration with ANSYS FLUENT computing the pull velocities, and then turn off the **Compute Pull Velocities** option and continue the calculation. For the remainder of the calculation, ANSYS FLUENT will use the values computed for the pull velocities at the first iteration.

27.3. Solution Procedure

Before solving the coupled fluid flow and heat transfer problem, you may want to patch an initial temperature or solve the steady conduction problem as an initial condition. The coupled problem can then be solved as either steady or transient. Because of the nonlinear nature of these problems, however, in most cases an transient solution approach is preferred.

You can specify the under-relaxation factor applied to the liquid fraction equation in the *Solution Controls Task Page* (p. 1902).

Solution Controls

Specify the desired value in the **Liquid Fraction Update** field under **Under-Relaxation Factors**. This sets the value of α_β in the following equation for updating the liquid fraction from one iteration (n) to the next ($n+1$):

$$\beta_{n+1} = \beta_n + \alpha_\beta \Delta\beta \tag{27–1}$$

where $\Delta\beta$ is the predicted change in liquid fraction.

In many cases, there is no need to change the default value of α_β. If, however, there are convergence difficulties, reducing the value may improve the solution convergence. Convergence difficulties can be expected in steady-state calculations, continuous casting simulations, simulations involving multicomponent solidification, and simulations where a large value of the mushy zone constant is used.

27.4. Postprocessing

For solidification/melting calculations, you can generate graphical plots or alphanumeric reports of the following items, which are all available in the **Solidification/Melting...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

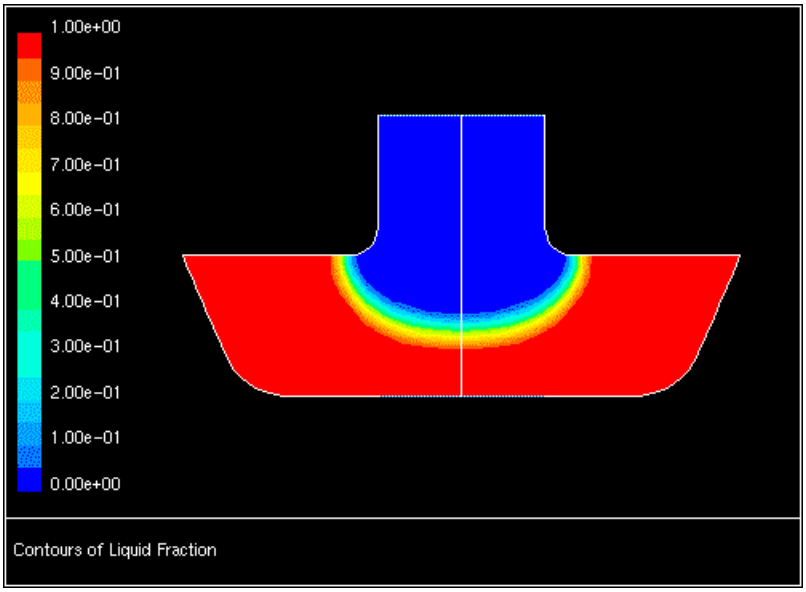
- **Liquid Fraction**
- **Contact Resistivity**
- **X**, **Y**, **Z**, **Axial**, **Radial**, or **Swirl Pull Velocity**
- **Liquidus Temperature**
- **Solidus Temperature**

The first two items are available for all solidification/melting simulations, and the others will appear only if you are including pull velocities (either computed or specified) in the simulation. See *Field Function Definitions* (p. 1529) or a complete list of field functions and their definitions. *Displaying Graphics* (p. 1379) and *Reporting Alphanumeric Data* (p. 1509) explain how to generate graphics displays and reports of data.

Figure 27.3 (p. 1206) shows filled contours of liquid fraction for a continuous crystal growth simulation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 27.3 Liquid Fraction Contours for Continuous Crystal Growth



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 28: Using the Solver

This chapter describes how to use the ANSYS FLUENT solver. For more information about the theory behind the ANSYS FLUENT solver, see "Solver Theory" in the Theory Guide. *Choosing the Solver* (p. 1209) provides an overview, and the remaining sections provide detailed instructions.

28.1. Overview of Using the Solver
28.2. Choosing the Spatial Discretization Scheme
28.3. Pressure-Based Solver Settings
28.4. Density-Based Solver Settings
28.5. Setting Algebraic Multigrid Parameters
28.6. Setting Solution Limits
28.7. Setting Multi-Stage Time-Stepping Parameters
28.8. Selecting Gradient Limiters
28.9. Initializing the Solution
28.10. Full Multigrid (FMG) Initialization
28.11. Hybrid Initialization
28.12. Performing Steady-State Calculations
28.13. Performing Pseudo Transient Calculations
28.14. Performing Time-Dependent Calculations
28.15. Monitoring Solution Convergence
28.16. Executing Commands During the Calculation
28.17. Automatic Initialization of the Solution and Case Modification
28.18. Animating the Solution
28.19. Checking Your Case Setup
28.20. Convergence and Stability
28.21. Solution Steering

28.1. Overview of Using the Solver

In ANSYS FLUENT, two solver technologies are available:

- pressure-based
- density-based

Both solvers can be used for a broad range of flows, but in some cases one formulation may perform better (i.e., yield a solution more quickly or resolve certain flow features better) than the other. The pressure-based and density-based approaches differ in the way that the continuity, momentum, and (where appropriate) energy and species equations are solved, as described in Overview of Flow Solvers in the Theory Guide.

The pressure-based solver traditionally has been used for incompressible and mildly compressible flows. The density-based approach, on the other hand, was originally designed for high-speed compressible flows. Both approaches are now applicable to a broad range of flows (from incompressible to highly compressible), but the origins of the density-based formulation may give it an accuracy (i.e. shock resolution) advantage over the pressure-based solver for high-speed compressible flows.

Two formulations exist under the density-based solver: implicit and explicit. The density-based explicit and implicit formulations solve the equations for additional scalars (e.g., turbulence or radiation quantities) se-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

quentially. The implicit and explicit density-based formulations differ in the way that they linearize the coupled equations. For more details about the solver formulations, see Overview of Flow Solvers in the Theory Guide.

Due to broader stability characteristics of the implicit formulation, a converged steady-state solution can be obtained much faster using the implicit formulation rather than the explicit formulation. However, the implicit formulation requires more memory than the explicit formulation.

Two algorithms also exist under the pressure-based solver in ANSYS FLUENT: a segregated algorithm and a coupled algorithm. In the segregated algorithm the governing equations are solved sequentially, segregated from one another, while in the coupled algorithm the momentum equations and the pressure-based continuity equation are solved in a coupled manner. In general, the coupled algorithm significantly improves the convergence speed over the segregated algorithm, however, the memory requirement for the coupled algorithm is more than the segregated algorithm.

When selecting a solver and an algorithm you must consider the following issues:

- The model availability for a given solver.
- Solver performance for the given flow conditions.
- The size of the mesh under consideration and the available memory on your machine. This issue could be an important factor in deciding whether to use an explicit or implicit formulation when the density-based solver is selected, or to use a segregated or coupled algorithm when the pressure-based solver is selected.

The following two lists highlight the model availability for each solver:

Important

Note that the pressure-based solver provides several physical models or features that are not available with the density-based solver:

- Cavitation model
- Volume-of-fluid (VOF) model
- Multiphase mixture model
- Eulerian multiphase model
- Non-premixed combustion model
- Premixed combustion model
- Partially premixed combustion model
- Composition PDF transport model
- Soot model
- Rosseland radiation model
- Melting/solidification model
- Shell conduction model
- Floating operating pressure
- Fixed variable option
- Physical velocity formulation for porous media

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specified mass flow rate for streamwise periodic flow

The following features are available with the density-based solver, but not with the pressure-based solver:

- Real gas models (User-defined and NIST)
- Non-reflecting boundary conditions
- Wet steam multiphase model

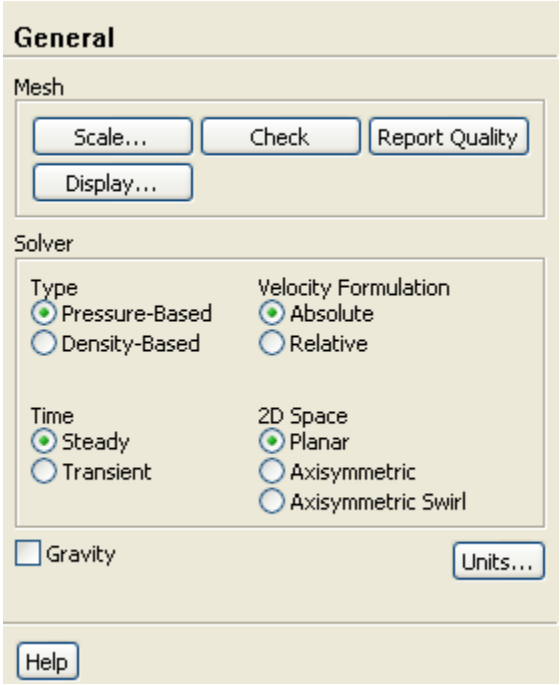
For additional information, please see the following sections:
28.1.1. Choosing the Solver

28.1.1. Choosing the Solver

To choose one of the solvers, you will use the *General Task Page* (p. 1631) (*Figure 28.1* (p. 1209)).

General

Figure 28.1 The General Task Page



To use the pressure-based solver, retain the default selection of **Pressure-Based** under **Solver**.

To use the density-based solver, select **Density-Based** under **Solver**.

After you have defined your model and specified which solver you want to use, you are ready to run the solver. The following steps outline a general procedure you can follow:

1. (pressure-based solver only) Select the pressure-velocity coupling method (see *Choosing the Pressure-Velocity Coupling Method* (p. 1215)).
2. Choose the spatial discretization scheme and, for the pressure-based solver, the pressure interpolation scheme (see *Choosing the Spatial Discretization Scheme* (p. 1210)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. (pressure-based solver only) Select the porous media velocity method (see *Porous Media Conditions* (p. 221)).
4. Select how you want the derivatives to be evaluated by choosing a gradient option (see Evaluation of Gradients and Derivatives in the Theory Guide).
5. Set the under-relaxation factors (see *Setting Under-Relaxation Factors* (p. 1217)).
6. (density-based explicit formulation only) Set up the FAS multigrid (see *Turning On FAS Multigrid* (p. 1225)).
7. Make any additional modifications to the solver settings that are suggested in the chapters or sections that describe the models you are using.
8. Enable the appropriate solution monitors (see *Monitoring Solution Convergence* (p. 1268)).
9. Initialize the solution (see *Initializing the Solution* (p. 1239)).
10. Start calculating (see *Performing Steady-State Calculations* (p. 1247) for steady state calculations, or *Performing Time-Dependent Calculations* (p. 1255) for time-dependent calculations).
11. If you have convergence trouble, try one of the methods discussed in *Convergence and Stability* (p. 1317).

The default settings for the first three items listed above are suitable for most problems and need not be changed. The following sections outline how these and other solution parameters can be changed, and when you may wish to change them.

28.2. Choosing the Spatial Discretization Scheme

Gradients are needed not only for constructing values of a scalar at the cell faces, but also for computing secondary diffusion terms and velocity derivatives. For more information about the different gradients, see Evaluation of Gradients and Derivatives in the Theory Guide.

The three gradients which are available in ANSYS FLUENT are

- **Green-Gauss Cell Based**
- **Green-Gauss Node Based**
- **Least Squares Cell Based**

The gradient options are selectable from the **Gradient** drop-down list, in the **Solution Methods** task page.

Solution Methods

In addition, ANSYS FLUENT allows you to choose the discretization scheme for the convection terms of each governing equation. (Second-order accuracy is automatically used for the viscous terms.) When the pressure-based solver is used, all equations are, by default, solved using the first-order upwind discretization for convection. When the density-based solver is used, the flow equations are solved using the second-order scheme by default, and the other equations use the first-order scheme by default. For a complete description of the discretization schemes available in ANSYS FLUENT, see Discretization in the Theory Guide.

In addition, when you use the pressure-based solver, you can specify the pressure interpolation scheme. For a description of the pressure interpolation schemes available in ANSYS FLUENT, see Pressure Interpolation Schemes in the Theory Guide.

For additional information, please see the following sections:

28.2.1. First-Order Accuracy vs. Second-Order Accuracy
28.2.2. Other Discretization Schemes
28.2.3. Choosing the Pressure Interpolation Scheme

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.2.4. Choosing the Density Interpolation Scheme
28.2.5. User Inputs

28.2.1. First-Order Accuracy vs. Second-Order Accuracy

When the flow is aligned with the mesh (e.g., laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral mesh) the first-order upwind discretization may be acceptable. When the flow is not aligned with the mesh (i.e., when it crosses the mesh lines obliquely), however, first-order convective discretization increases the numerical discretization error (numerical diffusion). For triangular and tetrahedral meshes, since the flow is never aligned with the mesh, you will generally obtain more accurate results by using the second-order discretization. For quad/hex meshes, you will also obtain better results using the second-order discretization, especially for complex flows.

In summary, while the first-order discretization generally yields better convergence than the second-order scheme, it generally will yield less accurate results, especially on tri/tet meshes. See *Convergence and Stability* (p. 1317) for information about controlling convergence.

For most cases, you will be able to use the second-order scheme from the start of the calculation. In some cases, however, you may need to start with the first-order scheme and then switch to the second-order scheme after a few iterations. For example, if you are running a high-Mach-number flow calculation that has an initial solution much different than the expected final solution, you will usually need to perform a few iterations with the first-order scheme and then turn on the second-order scheme and continue the calculation to convergence. Alternatively, full multigrid initialization is also available for some flow cases which allow you to proceed with the second-order scheme from the start.

For a simple flow that is aligned with the mesh (e.g., laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral mesh), the numerical diffusion will be naturally low, so you can generally use the first-order scheme instead of the second-order scheme without any significant loss of accuracy.

Finally, if you run into convergence difficulties with the second-order scheme, you should try the first-order scheme instead.

28.2.1.1. First-to-Higher Order Blending

While the higher-order scheme may result in greater accuracy, it can also result in convergence difficulties and instabilities at certain flow conditions. On the other hand, using a first-order scheme may not provide the desired accuracy. One approach to achieving improved accuracy while maintaining good stability is to use a discretization blending factor. This feature is available for both density-based and pressure-based solvers and can be invoked using the following text command:

`solve` → `set` → `numerics`

Enter a value between `0` and `1`when asked for the blending factor: `1st-order to higher-order blending factor [min=0.0 - max=1.0]`

A blending factor of `0` reduces the gradient reconstruction to a first-order discretization scheme, whereas `1` will recover high-order discretization. A blending factor of less than `1` (typically 0.75 or 0.5) will make the convective fluxes more diffusive, which in some flow conditions can stabilize a solution that is otherwise unstable when the full higher-order discretization scheme is employed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that in order to use this feature effectively, make sure that one of the allowed higher order discretization schemes is selected for the desired variables in the **Solution Methods** task page.

28.2.2. Other Discretization Schemes

The QUICK and third-order MUSCL discretization schemes may provide better accuracy than the second-order scheme for rotating or swirling flows. The QUICK scheme is applicable to quadrilateral or hexahedral meshes, while the MUSCL scheme is used on all types of meshes. In general, however, the second-order scheme is sufficient and the QUICK scheme will not provide significant improvements in accuracy.

Important

If QUICK is used for hybrid meshes, it will be used only for quadrilateral and hexahedral cells. Second-order upwind discretization will be applied to all other cells.

A power law scheme is also available, but it will generally yield the same accuracy as the first-order scheme.

The bounded central differencing and central differencing schemes are available only when you are using the LES and DES turbulence models, and the central differencing scheme should be used only when the mesh spacing is fine enough so that the magnitude of the local Peclet number (see Equation 19–6 in the Theory Guide) is less than 1.

A modified HRIC scheme (see Modified HRIC Scheme in the Theory Guide) is also available for VOF simulations using either the implicit or explicit formulation.

28.2.3. Choosing the Pressure Interpolation Scheme

As discussed in Pressure Interpolation Schemes in the Theory Guide, a number of pressure interpolation schemes are available when the pressure-based solver is used in ANSYS FLUENT. For most cases the "standard" scheme is acceptable, but some types of models may benefit from one of the other schemes:

- For problems involving large body forces, the body-force-weighted scheme is recommended.
- For flows with high swirl numbers, high-Rayleigh-number natural convection, high-speed rotating flows, flows involving porous media, and flows in strongly curved domains, use the PRESTO! scheme.
- For compressible flows, the second-order scheme is recommended.
- Use the second-order scheme for improved accuracy when one of the other schemes is not applicable.

Important

The second-order scheme cannot be used with porous media or porous jump.

Important

Only PRESTO! and body-force-weighted schemes are available for the VOF model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you will not specify the pressure interpolation scheme if you are using the Eulerian multiphase model. ANSYS FLUENT will use the solution method described in Solution Method in ANSYS FLUENT in the Theory Guide for Eulerian multiphase calculations.

28.2.4. Choosing the Density Interpolation Scheme

As discussed in Density Interpolation Schemes in the Theory Guide, four density interpolation schemes are available when the pressure-based solver is used to solve a single-phase compressible flow.

The first-order upwind scheme (the default) provides stability for the discretization of the pressure-correction equation, and gives good results for most classes of flows. If you are calculating a compressible flow with shocks, the first-order upwind scheme may tend to smooth the shocks; you should use the second-order-upwind or QUICK scheme for such flows. For compressible flows with shocks, using the QUICK scheme for all variables, including density, is highly recommended for quadrilateral, hexahedral, or hybrid meshes. The third-order MUSCL scheme is applicable to arbitrary meshes and has the potential to improve spatial accuracy for all types of meshes by reducing numerical diffusion.

Important

In the case of multiphase flows, the selected density scheme is applied to the compressible phase and arithmetic averaging is used for incompressible phases.

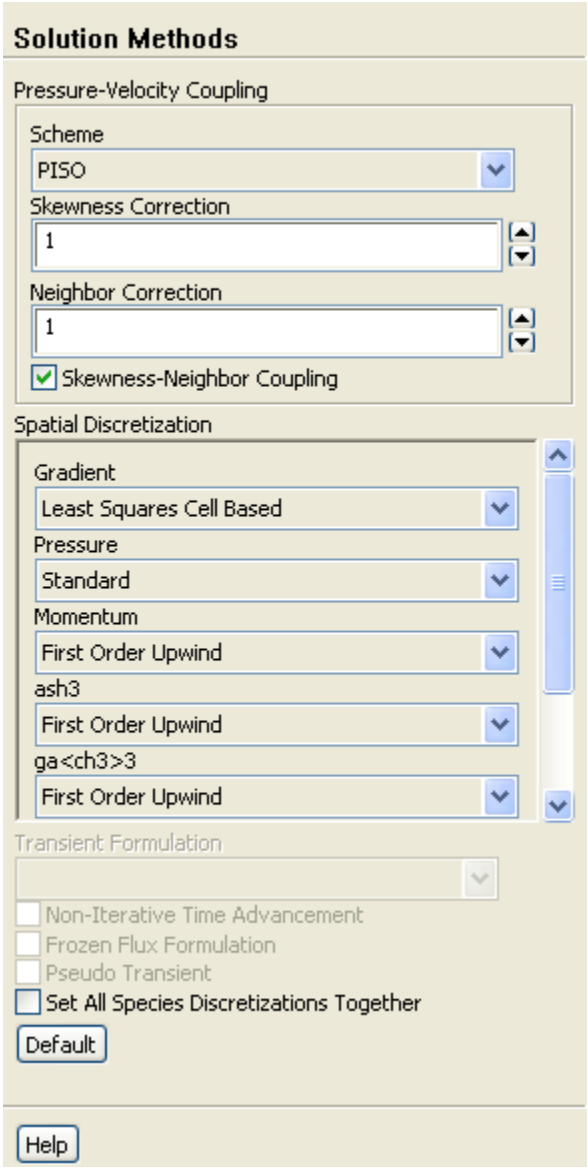
28.2.5. User Inputs

You can specify the discretization scheme and, for the pressure-based solver, the pressure interpolation scheme in the *Solution Methods Task Page* (p. 1900) (*Figure 28.2* (p. 1214)).

Solution Methods

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.2 The Solution Methods Task Page for the Pressure-Based Segregated Algorithm



For each scalar equation listed under **Spatial Discretization** (**Momentum**, **Energy**, **Turbulent Kinetic Energy**, etc. for the pressure-based solver or **Turbulent Kinetic Energy**, **Turbulent Dissipation Rate**, etc. for the density-based solver) you can choose **First Order Upwind**, **Second Order Upwind**, **Power Law**, **QUICK**, **Third-Order MUSCL**, or (if you are using the LES turbulence model) **Bounded Central Differencing** (the default) or **Central Differencing** in the adjacent drop-down list. For the density-based solver, you can choose either **First Order Upwind**, **Second Order Upwind**, or **Third-Order MUSCL** for the **Flow** equations (which include momentum and energy). Note that the task page shown in *Figure 28.2* (p. 1214) is for the pressure-based solver.

If you are using the pressure-based solver, select the pressure interpolation scheme under **Spatial Discretization**, in the drop-down list next to **Pressure**. You can choose **Standard**, **PRESTO!**, **Linear**, **Second Order**, or **Body Force Weighted**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The low order modification of PRESTO! can be applied by disabling the high order terms for the PRESTO! scheme. This is done using the following text command:

`solve` → `set` → `numerics`

When asked `disable high order terms for PRESTO! pressure scheme?`, enter `yes`.

This modification can be used to stabilize the solution process when the pressure-based coupled algorithm is used and when the original PRESTO! scheme fails to converge.

If you are using the pressure-based solver and your flow is compressible (i.e., you are using the ideal gas law for density), select the density interpolation scheme under **Spatial Discretization**, in the drop-down list next to **Density**. You can choose **First Order Upwind**, **Second Order Upwind**, **QUICK** or **Third-Order MUSCL**. (Note that **Density** will not appear for incompressible flows.)

If you enable the VOF model while using the pressure-based solver, the volume fraction interpolation schemes that are available are **Geo-Reconstruct**, **CICSAM**, **Modified HRIC**, and **QUICK**.

If your case involves species transport, you can set the scheme for the individual species as **First Order Upwind**, **Second Order Upwind**, **Power Law**, **QUICK**, or **Third-Order MUSCL**. However, if you want all your species to use the same discretization scheme, then rather than setting each one individually, simply enable the **Set All Species Discretizations Together** option. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the scheme of your choice.

If you change the settings for the **Spatial Discretization**, but you then want to return to ANSYS FLUENT's default settings, you can click the **Default** button.

28.3. Pressure-Based Solver Settings

For additional information, please see the following sections:

28.3.1. Choosing the Pressure-Velocity Coupling Method
28.3.2. Setting Under-Relaxation Factors
28.3.3. Setting Solution Controls for the Non-Iterative Solver

28.3.1. Choosing the Pressure-Velocity Coupling Method

ANSYS FLUENT provides four segregated types of algorithms: SIMPLE, SIMPLEC, PISO, and (for time-dependant flows using the **Non-Iterative Time Advancement** option (NITA)) Fractional Step (FSM). These schemes are referred to as the pressure-based segregated algorithm. Steady-state calculations will generally use SIMPLE or SIMPLEC, while PISO is recommended for transient calculations. PISO may also be useful for steady-state and transient calculations on highly skewed meshes. In ANSYS FLUENT, using the **Coupled** algorithm enables full pressure-velocity coupling, hence it is referred to as the pressure-based coupled algorithm.

Important

Pressure-velocity coupling is relevant only for the pressure-based solver.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.3.1.1. SIMPLE vs. SIMPLEC

In ANSYS FLUENT, both the standard SIMPLE algorithm and the SIMPLEC (SIMPLE-Consistent) algorithm are available. SIMPLE is the default, but many problems will benefit from using SIMPLEC, particularly because of the increased under-relaxation that can be applied, as described below.

For relatively uncomplicated problems (laminar flows with no additional models activated) in which convergence is limited by the pressure-velocity coupling, you can often obtain a converged solution more quickly using SIMPLEC. With SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which aids in convergence speed-up. In some problems, however, increasing the pressure-correction under-relaxation to 1.0 can lead to instability due to high mesh skewness. For such cases, you will need to use one or more skewness correction schemes, use a slightly more conservative under-relaxation value (up to 0.7), or use the SIMPLE algorithm. For complicated flows involving turbulence and/or additional physical models, SIMPLEC will improve convergence only if it is being limited by the pressure-velocity coupling. Often it will be one of the additional modeling parameters that limits convergence; in this case, SIMPLE and SIMPLEC will give similar convergence rates.

28.3.1.2. PISO

The PISO algorithm (see PISO in the Theory Guide) with neighbor correction is highly recommended for all transient flow calculations, especially when you want to use a large time step. (For problems that use the LES turbulence model, which usually requires small time steps, using PISO may result in an increased computational expense, so SIMPLE or SIMPLEC should be considered instead.) PISO can maintain a stable calculation with a larger time step and an under-relaxation factor of 1.0 for both momentum and pressure. For steady-state problems, PISO with neighbor correction does not provide any noticeable advantage over SIMPLE or SIMPLEC with optimal under-relaxation factors.

PISO with skewness correction is recommended for both steady-state and transient calculations on meshes with a high degree of distortion.

When you use PISO neighbor correction, under-relaxation factors of 1.0 or near 1.0 are recommended for all equations. If you use just the PISO skewness correction for highly-distorted meshes (without neighbor correction), set the under-relaxation factors for momentum and pressure so that they sum to 1 (e.g., 0.3 for pressure and 0.7 for momentum). If you use both PISO methods, follow the under-relaxation recommendations for PISO neighbor correction, above.

For most problems, it is not necessary to disable the default coupling between neighbor and skewness corrections. For highly distorted meshes, however, disabling the default coupling between neighbor and skewness corrections is recommended.

28.3.1.3. Fractional Step Method

The Fractional Step method (FSM), described in Fractional-Step Method (FSM) in the Theory Guide, is available when you choose to use the NITA scheme (i.e., the **Non-Iterative Time Advancement** option in the **Solution Methods** task page). With the NITA scheme, the FSM is slightly less computationally expensive compared to the PISO algorithm. Whether you select FSM or PISO depends on the application. For some problems (e.g., simulations that use VOF), FSM could be less stable than PISO.

In most cases, the default values for the solution methods are enough to set a robust convergence of the internal pressure correction sub-iterations due to skewness. Only very complex problems (e.g., moving deforming meshes, sliding interfaces, the VOF model) could require a reduction of relaxation for pressure up to a value of 0.7 or 0.8.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.3.1.4. Coupled

Selecting **Coupled** from the **Pressure-Velocity Coupling** drop-down list indicates that you are using the pressure-based coupled algorithm, described in Coupled Algorithm in the Theory Guide. This solver offers some advantages over the pressure-based segregated algorithm. The pressure-based coupled algorithm obtains a more robust and efficient single phase implementation for steady-state flows. It is not available for cases using the Eulerian multiphase, NITA, and periodic mass-flow boundary conditions.

28.3.1.4.1. User Inputs

You can specify the pressure-velocity coupling method in the *Solution Methods Task Page* (p. 1900) (*Figure 28.2* (p. 1214)).

Solution Methods

Choose **SIMPLE**, **SIMPLEC**, **PISO**, **Fractional Step**, or **Coupled** in the **Pressure-Velocity Coupling** drop-down list.

If you choose **PISO**, the task page will expand to show the additional parameters for pressure-velocity coupling. By default, the number of iterations for **Skewness Correction** and **Neighbor Correction** are set to `1`. If you want to use only **Skewness Correction**, then set the number of iterations for **Neighbor Correction** to `0`. Likewise, if you want to use only **Neighbor Correction**, then set the number of iterations for **Skewness Correction** to `0`. For most problems, you do not need to change the default iteration values. By default, the **Skewness-Neighbor Coupling** option is enabled to allow for a more economical, but a less robust variation of the PISO algorithm.

If you choose **SIMPLEC** under **Pressure-Velocity Coupling**, you must also set the **Skewness Correction**, whose default value is `0`.

If you choose **Coupled**, you will have to specify the **Courant** number in the **Solution Controls** task page, which is set at `200` by default. You will also specify the **Explicit Relaxation Factors** for **Momentum** and **Pressure**, which are set at `0.75` by default. For more information about these options, refer to Pressure-Velocity Coupling and Steady-State Iterative Algorithm in the Theory Guide.

If high-order momentum discretization is used, you may need to decrease the explicit relaxation to `0.5`. For cases with very skewed meshes, the run can be stabilized by further reduction of the explicit relaxation factor to `0.25`. If ANSYS FLUENT immediately diverges in the AMG solver, then the CFL number is too high and should be reduced. Reducing the CFL number below `10` is not recommended since it would be better to use the segregated algorithm for the pressure-velocity coupling.

In most transient cases, the CFL number should be set to 10^7 with an explicit relaxation of 1.0.

If you choose **Coupled** and enable the **Pseudo Transient** option, you will set the **Pseudo Transient Explicit Relaxation Factors** in the **Solution Controls** task page, as described in *Setting Pseudo Transient Explicit Relaxation Factors* (p. 1250).

28.3.2. Setting Under-Relaxation Factors

The pressure-based solver uses under-relaxation of equations to control the update of computed variables at each iteration (as described in Under-Relaxation of Equations in the Theory Guide). This means that all equations solved using the pressure-based solver, *including the non-coupled equations solved by the density-based solver* (turbulence and other scalars, as discussed in Density-Based Solver in the Theory Guide), will have under-relaxation factors associated with them.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In ANSYS FLUENT, the default under-relaxation parameters for all variables are set to values that are near optimal for the largest possible number of cases. These values are suitable for many problems, but for some particularly nonlinear problems (e.g., some turbulent flows or high-Rayleigh-number natural-convection problems) it is prudent to reduce the under-relaxation factors initially.

It is good practice to begin a calculation using the default under-relaxation factors. If the residuals continue to increase after the first 4 or 5 iterations, you should reduce the under-relaxation factors.

Occasionally, you may make changes in the under-relaxation factors and resume your calculation, only to find that the residuals begin to increase. This often results from increasing the under-relaxation factors too much. A cautious approach is to save a data file before making any changes to the under-relaxation factors, and to give the solution algorithm a few iterations to adjust to the new parameters. Typically, an increase in the under-relaxation factors brings about a slight increase in the residuals, but these increases usually disappear as the solution progresses. If the residuals jump by a few orders of magnitude, you should consider halting the calculation and returning to the last good data file saved.

Note that viscosity and density are under-relaxed from iteration to iteration. Also, if the enthalpy equation is solved directly instead of the temperature equation (i.e., for non-premixed combustion calculations), the update of temperature based on enthalpy will be under-relaxed. To see the default under-relaxation factors, you can click the **Default** button in the *Solution Controls Task Page* (p. 1902).

For most flows, the default under-relaxation factors do not usually require modification. If unstable or divergent behavior is observed, however, you need to reduce the under-relaxation factors for pressure, momentum, k, and ε from their default values to about 0.2, 0.5, 0.5, and 0.5. (It is usually not necessary to reduce the pressure under-relaxation for SIMPLEC.) In problems where density is strongly coupled with temperature, as in very-high-Rayleigh-number natural- or mixed-convection flows, it is wise to also under-relax the temperature equation and/or density (i.e., use an under-relaxation factor less than 1.0). Conversely, when temperature is not coupled with the momentum equations (or when it is weakly coupled), as in flows with constant density, the under-relaxation factor for temperature can be set to 1.0.

For other scalar equations (e.g., swirl, species, mixture fraction and variance) the default under-relaxation may be too aggressive for some problems, especially at the start of the calculation. You may wish to reduce the factors to 0.8 to facilitate convergence.

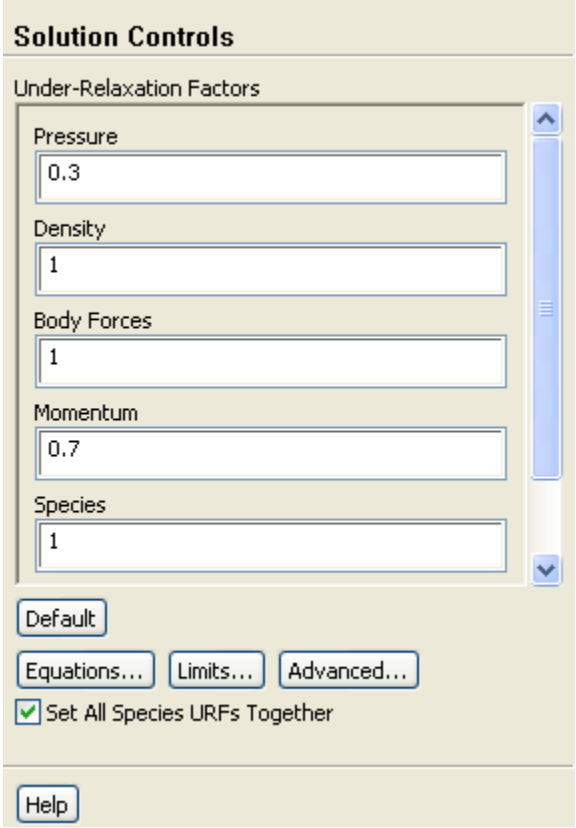
28.3.2.1. User Inputs

You can modify the under-relaxation factors in the *Solution Controls Task Page* (p. 1902) (*Figure 28.3* (p. 1219)).

Solution Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.3 The Solution Controls Task Page for the Pressure-Based Solver



You can set the under-relaxation factor for each equation in the field next to its name under **Under-Relaxation Factors**.

Important

If you are using the pressure-based solver, all equations will have an associated under-relaxation factor (see Under-Relaxation of Equations in the Theory Guide). If you are using the density-based solver, only those equations that are solved sequentially (see Density-Based Solver in the Theory Guide) will have under-relaxation factors.

If your case involves species transport, you can set the under-relaxation factors for each of the listed species. If you want all your species to use the same under-relaxation factors, simply enable the **Set All Species URFs Together** option. Notice that you will no longer see your list of individual species, instead a **Species** field will appear where you will specify the under-relaxation factor.

If you change under-relaxation factors, but you then want to return to ANSYS FLUENT's default settings, you can click the **Default** button.

Note that with optimal settings, the convergence of the coupled pressure-velocity algorithm will be limited by the segregated solution of other scalar equations, e.g., turbulence. For optimum solver performance, you will need to increase the relaxation factors for these equations to a value greater than the default values.

28.3.3. Setting Solution Controls for the Non-Iterative Solver

You can use the non-iterative solver (see Time-Advancement Algorithm in the Theory Guide) for transient problems in order to increase the speed and efficiency of the calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The settings for the non-iterative solver should provide control over the maximum number of sub-iterations for each individual equation. The criteria for convergence include the **Correction Tolerance** (defined by the overall accuracy), **Residual Tolerance** (controlling the solution of the linear equations), and the individual **Relaxation Factor**. The default control settings are optimally designed in order to get a second-order accurate solution. These controls are accessible via the **Expert** tab, in the **Advanced Solution Controls** dialog box (*Figure 28.4* (p. 1220)).

To use ANSYS FLUENT's non-iterative transient solver in order to boost the efficiency of transient simulations:

1. Go to the **Solution Methods** task page.

   Solution Methods

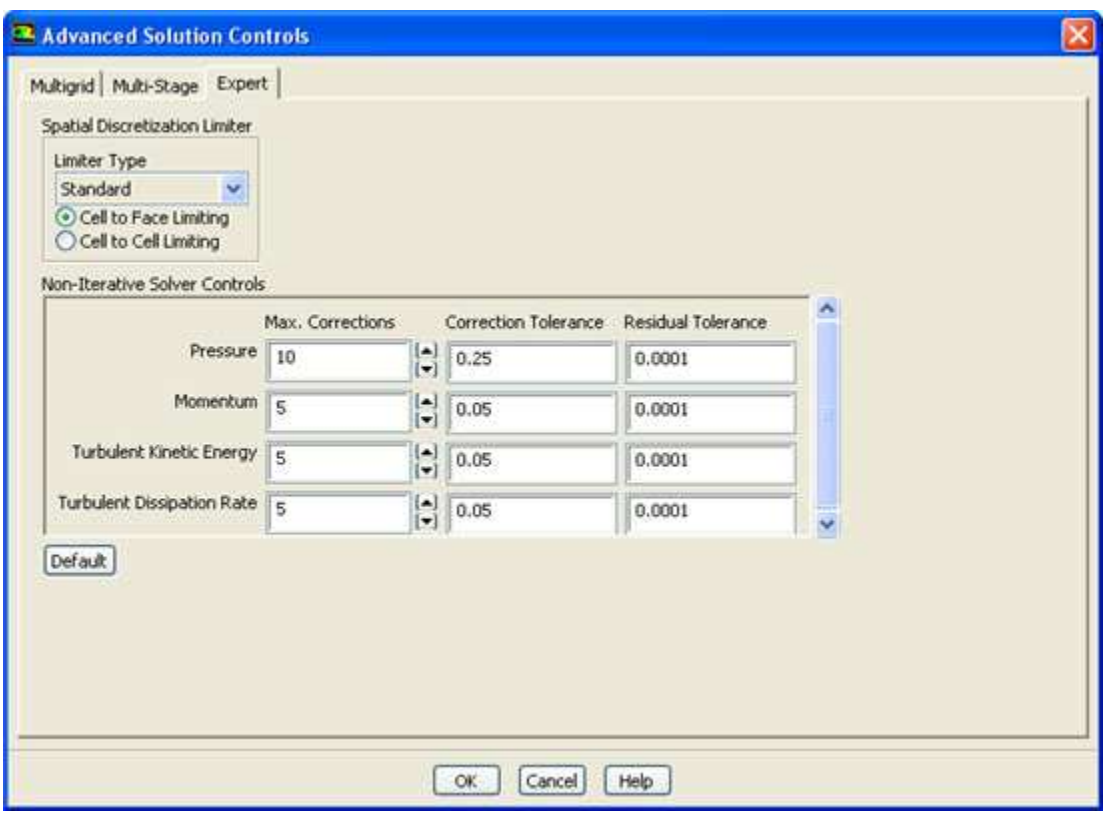
2. Enable **Non-Iterative Time-Advancement**.
3. Under **Pressure-Velocity Coupling**, you can choose either the **Fractional Step** or **PISO** scheme. Under **Non-Iterative Solver Controls** (*Figure 28.4* (p. 1220)), you will see parameters that control the sub-iterations for individual equations (see below). When you select the PISO scheme, you can set the value for the **Neighbor Correction**. Skewness correction is performed automatically.

28.3.3.1. User Inputs

You can modify the non-iterative solution controls in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.4* (p. 1220)).

Solution Controls → **Advanced...**

Figure 28.4 The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under **Non-Iterative Solver Controls**, there are several parameters that control the sub-iterations for the individual equations.

The sub-iterations for an equation stop when the total number of sub-iterations exceeds the value specified for **Max. Corrections**, regardless of whether or not the convergence criteria (described below) are met.

The sub-iterations for an equation end when the ratio of the residuals at the current sub-iteration and the first sub-iteration is less than the value specified in the **Correction Tolerance** field. You can monitor the details of the sub-iteration convergence by looking at the AMG solver performance (i.e., setting the **Verbosity** field in the **Multigrid** tab in the **Advanced Solution Controls** dialog box to **1**). Be sure to pay attention to the residuals for the current sub-iteration (i.e., the residual for the 0-th AMG cycle at the current sub-iteration) and the initial residual of the time step (i.e., the residual for the 0-th AMG cycle of the first sub-iteration). The ratio of these two residuals is what is controlled by the **Correction Tolerance** field. These two residuals are also the residuals plotted when using the **Residual Monitor** panel and reported in the ANSYS FLUENT console at the end of a time step. Note that the residuals reported at the end of a time step can be scaled or unscaled, depending on the settings in the **Residual Monitor** dialog box. The residuals reported when monitoring the AMG solver performance are always unscaled.

For each interim sub-iteration, the AMG cycles continue until the usual AMG termination criteria (0.1 by default, and set in the **Multigrid** tab) are met. However, for the last sub-iteration (i.e., either when the maximum number of sub-iterations are reached or when the correction tolerance is satisfied), the AMG cycles continue until the ratio of the residual at the current cycle to the initial residual (the residual for the 0-th AMG cycle of the first sub-iteration of the time step) drops below the value specified for **Residual Tolerance**. You may want to adjust the **Residual Tolerance**, depending on the time step selected. The default **Residual Tolerance** should be well suited for moderate time steps (i.e., for cell CFL numbers of 1 to 10). Note that you can display the cell CFL numbers for unsteady problems by selecting **Cell Courant Number** in the **Velocity...** category of all postprocessing dialog boxes. For very small time steps (cell CFL <<1), the diagonal dominance of the system is very high and the convergence should be driven further by reducing the **Residual Tolerance** value. For larger time steps (cell CFL >>1), it may be possible that the residual tolerance cannot be reached due to round-off errors, and unless the **Residual Tolerance** value is increased, AMG cycles can be wasted. Again, this can be monitored by monitoring the AMG solver performance.

The **Relaxation Factor** field defines the explicit relaxation (see Under-Relaxation of Variables in the Theory Guide) of variables between sub-iterations. The relaxation factors can be used to prevent the solution from diverging. They should be left at their default values of `1`, unless divergence is detected. If the solution diverges, you should first try to stabilize the solution by lowering the relaxation factors for pressure to `0.7`–`0.8`, and by reducing the time step.

The following is a list of models that are compatible with the non-iterative solver:

- Inviscid flow (excluding ideal gas)
- Laminar flow
- All models of turbulence (including LES and DES), except RSM
- S2S radiation model
- Heat transfer
- Non-reacting species transport
- General compressible flows (most subsonic and some transonic applications)
- VOF multiphase model (most applications)
- Phase change (solidification and melting)
- Porous media model (isotropic resistance)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is a list of models that are compatible with the non-iterative solver, but may result in some instabilities and inaccuracies for certain flow conditions:

- MDM
- Non-Newtonian fluids
- General compressible flows (aerospace supersonic applications)
- Floating operating pressure
- Reacting species and any type of combustion including PDF

The following is a list of models that are not compatible with the non-iterative solver:

- Eulerian multiphase (all non-VOF models)
- Radiation models (except S2S)
- DPM, spark, and crevice models
- UDS transport
- Porous jump
- Porous media model (anisotropic resistance)
- RSM turbulence model

Important

The PRESTO! pressure interpolation scheme, when used with the non-iterative time-advancement solver, is less stable than in the case of the iterative time-advancement solver. As a consequence, smaller time steps may be required.

Important

As mentioned above, the default control settings are optimally designed to obtain a second-order solution. In order to save CPU time, in cases where transient accuracy is not a main concern (i.e., first-order integration in time and space), or when NITA is used to converge toward a steady state solution, you may want to set the **Max. Corrections** value to 1 in the **Advanced Solution Controls** dialog box (**Expert** tab) for all transport equations except pressure.

28.4. Density-Based Solver Settings

For additional information, please see the following sections:

28.4.1. Changing the Courant Number
28.4.2. Convective Flux Types
28.4.3. Specifying the Explicit Relaxation
28.4.4. Turning On FAS Multigrid

28.4.1. Changing the Courant Number

For ANSYS FLUENT's density-based solver, the main control over the time-stepping scheme is the Courant number (CFL). The time step is proportional to the CFL, as defined in Equation 19–79 in the Theory Guide.

Linear stability theory determines a range of permissible values for the CFL (i.e., the range of values for which a given numerical scheme will remain stable). When you specify a permissible CFL value, ANSYS FLUENT will

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

compute an appropriate time step using Equation 19–79 in the Theory Guide. In general, taking larger time steps leads to faster convergence, so it is advantageous to set the CFL as large as possible (within the permissible range).

The stability limits of the density-based implicit and explicit formulations are significantly different. The explicit formulation has a more limited range and requires lower CFL settings than does the density-based implicit formulation. Appropriate choices of CFL for the two formulations are discussed below.

28.4.1.1. Courant Numbers for the Density-Based Explicit Formulation

Linear stability analysis shows that the maximum allowable CFL for the multi-stage scheme used in the density-based explicit formulation will depend on the number of stages used and how often the dissipation and viscous terms are updated (see *Changing the Multi-Stage Scheme* (p. 1237)). But in general, you can assume that the multi-stage scheme is stable for Courant numbers up to 2.5. This stability limit is often lower in practice because of nonlinearities in the governing equations.

The default CFL for the density-based explicit formulation is 1.0, but you may be able to increase it for some 2D problems. You should generally not use a value higher than 2.0.

If your solution is diverging, i.e., if residuals are rising very rapidly, and your problem is properly set up and initialized, this is usually a good sign that the Courant number needs to be lowered. Depending on the severity of the startup conditions, you may need to decrease the CFL to a value as low as 0.1 to 0.5 to get started. Once the startup transients are reduced you can start increasing the Courant number again.

28.4.1.2. Courant Numbers for the Density-Based Implicit Formulation

Linear stability theory shows that the density-based implicit formulation is unconditionally stable. However, as with the explicit formulation, nonlinearities in the governing equations will often limit stability.

The default CFL for the density-based implicit formulation is 5.0. It is often possible to increase the CFL to 10, 20, 100, or even higher, depending on the complexity of your problem. You may find that a lower CFL is required during startup (when changes in the solution are highly nonlinear), but it can be increased as the solution progresses.

The coupled AMG solver has the capability to detect divergence of the multigrid cycles within a given iteration. If this happens, it will automatically reduce the CFL and perform the iteration again, and a message will be printed to the screen. Five attempts are made to complete the iteration successfully. Upon successful completion of the current iteration the CFL is returned to its original value and the iteration procedure proceeds as required.

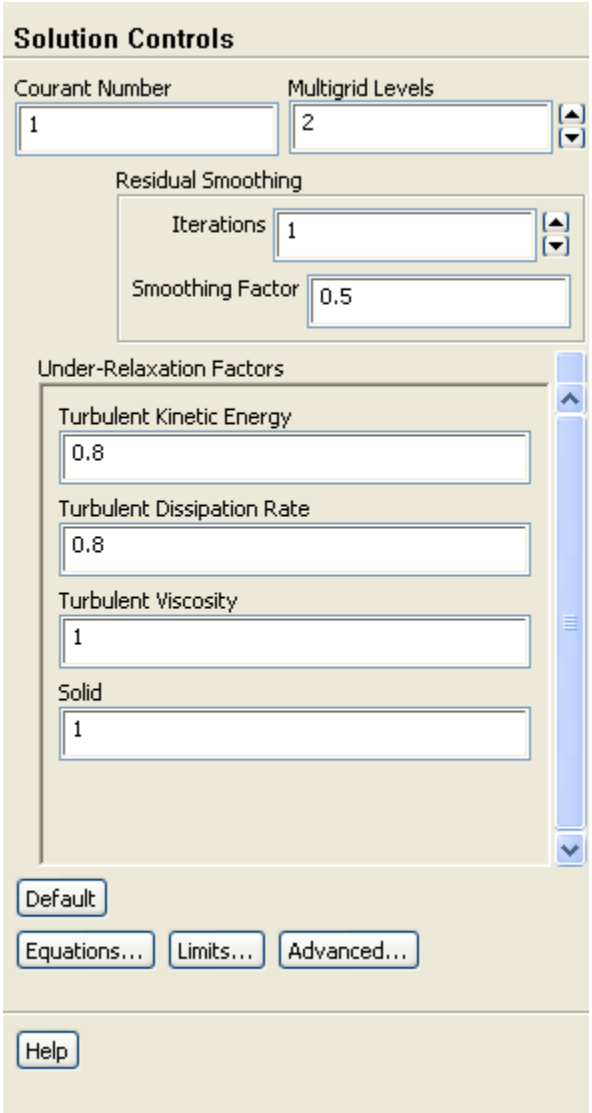
28.4.1.3. User Inputs

The Courant number is set in the *Solution Controls Task Page* (p. 1902) (*Figure 28.5* (p. 1224)).

Solution Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.5 The Solution Controls Task Page for the Density-Based Explicit Formulation



Enter the value for **Courant Number**. (Note that the task page shown in *Figure 28.5* (p. 1224)

When you select **Explicit** from the **Formulation** drop-down list, in the *Solution Methods Task Page* (p. 1900), ANSYS FLUENT will automatically set the **Courant Number** to 1; when you select **Implicit** from the **Formulation** drop-down list, the **Courant Number** will be changed to 5 automatically.

28.4.2. Convective Flux Types

Three convective flux types exist when using the density-based solver:

- Roe flux-difference splitting (Roe-FDS)
- Advection Upstream Splitting Method (AUSM)
- Low diffusion Roe flux-difference splitting (Low Diffusion Roe-FDS)

Roe-FDS splits the fluxes in a manor that is consistent with their corresponding flux method eigenvalues. It is the default and is recommended for most cases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUSM provides exact resolution of contact and shock discontinuities and it is less susceptible to Carbuncle phenomena.

Low diffusion Roe-FDS is available in special circumstances when the LES viscous model is enabled and when time-implicit formulation is used in the density-based solvers. It reduces the dissipation in LES calculations. Low diffusion Roe-FDS should be used only for subsonic flows.

28.4.2.1. User Inputs

The convective fluxes are selected from the **Flux Type** drop-down list in the **Solution Methods** task page.

Solution Methods

Select **Roe-FDS**, **AUSM**, or if the LES viscous model is enabled with the time-implicit formulation, **Low Diffusion Roe-FDS** will be available.

28.4.3. Specifying the Explicit Relaxation

To improve the convergence to steady state for some flow cases when using the density-based implicit solver you can specify the explicit relaxation using the following text command:

`solve` → `set` → `expert`

Enter a value between `0` and `1` in response to the `Explicit relaxation value` prompt.

For more information about explicit relaxation, see Under-Relaxation of Variables in the Theory Guide.

28.4.4. Turning On FAS Multigrid

As discussed in Multigrid Method in the Theory Guide, FAS multigrid is an optional component of the density-based explicit formulation, while AMG multigrid is always on, by default for the density-based implicit formulation. Since nearly all density-based explicit calculations will benefit from the use of the FAS multigrid convergence accelerator, you should generally set a non-zero number of coarse grid levels before beginning the calculation. For most problems, this will be the only FAS multigrid parameter you will need to set. Should you encounter convergence difficulties, consider applying one of the methods discussed in *Setting FAS Multigrid Parameters* (p. 1232).

Important

Note that you cannot use FAS multigrid with explicit time stepping (described in Temporal Discretization in the Theory Guide) because the coarse grid corrections will destroy the time accuracy of the fine grid solution.

28.4.4.1. Setting Coarse Grid Levels

As discussed in Full-Approximation Storage (FAS) Multigrid in the Theory Guide, FAS multigrid solves on successively coarser grids and then transfers corrections to the solution back up to the original fine grid, thus increasing the propagation speed of the solution and speeding convergence. The most basic way you can control the multigrid solver is by specifying the number of coarse grid levels to be used.

As explained in Full-Approximation Storage (FAS) Multigrid in the Theory Guide, the coarse grid levels are formed by agglomerating a group of adjacent “fine” cells into a single “coarse” cell. The optimal number of

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

grid levels is therefore problem-dependent. For most problems, you can start out with 4 or 5 levels. For large 3D problems, you may want to add more levels (although memory restrictions may prevent you from using more levels, since each coarse grid level requires additional memory). If you believe that multigrid is causing convergence trouble, you can decrease the number of levels.

If ANSYS FLUENT reaches a coarse grid with one cell before creating as many levels as you requested, it will simply stop there. That is, if you request 5 levels, and level 4 has only 1 cell, ANSYS FLUENT will create only 4 levels, since levels 4 and 5 would be the same.

To specify the number of grid levels you want, set the number of **Multigrid Levels** in the *Solution Controls Task Page* (p. 1902) (*Figure 28.5* (p. 1224)).

Solution Controls

You can also set the **Max Coarse Levels** under **FAS Multigrid Controls** in the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906).

Changing the number of coarse grid levels in the **Solution Controls** task page will automatically update the number shown in the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906).

Coarse grid levels are created when you first begin iterating. If you want to check how many cells are in each level, request one iteration and then use the **Mesh/Info/Size** menu item (described in *Mesh Size* (p. 175)) to list the size of each grid level. If you are satisfied, you can continue the calculation; if not, you can change the number of coarse grid levels and check again.

For most problems, you will not need to modify any additional multigrid parameters once you have settled on an appropriate number of coarse grid levels. You can simply continue your calculation until convergence.

28.4.4.2. Using Residual Smoothing to Increase the Courant Number

In the density-based explicit formulation, implicit residual smoothing (or averaging) is a technique that can be used to reduce the time step restriction of the solver, thereby allowing the Courant number to be increased. The implicit smoothing is implemented with an iterative Jacobi method, as described in Implicit Residual Smoothing in the Theory Guide. *Solution Controls Task Page* (p. 1902).

Solution Controls

By default, the number of **Iterations** for **Residual Smoothing** is set to zero, indicating that residual smoothing is disabled. If you increase the **Iterations** counter to 1 or more, you can enter the **Smoothing Factor**. A smoothing factor of 0.5 with 2 passes of the Jacobi smoother is usually adequate to allow the Courant number to be doubled.

28.5. Setting Algebraic Multigrid Parameters

As mentioned earlier, in most cases the multigrid solver will not require any special attention from you. If, however, you have convergence difficulties or you want to minimize the overall solution time by using more aggressive settings, you can monitor the multigrid solver and modify the parameters to improve its performance. (The instructions below assume that you have already begun calculations, since there is no need to monitor the solver if you do not fit into one of the two categories above.)

To determine whether your convergence difficulties can be alleviated by modifying the multigrid settings, you will check if the requested residual reduction is obtained on each grid level. To minimize solution time,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you will check to see if switching to a more powerful cycle will result in overall reduction of work by the solver.

By default, the flexible cycle is used for all equations except pressure correction, which uses a V cycle. Typically, for a flexible cycle only a few (5–10) relaxations will be performed at the finest level and no coarse levels will be visited. In some cases one or two coarse levels may be visited. If the maximum number of fine level relaxations is not sufficient, you may want to increase the maximum number (as described in *Flexible Cycle Parameters* (p. 1230)) or switch to a V cycle (as described in *Specifying the Multigrid Cycle Type* (p. 1227)).

In the pressure-based segregated algorithm, the pressure correction uses a V cycle by default. If the maximum number of cycles (30 by default) is not sufficient, you can switch to a W cycle (using the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906), as described in *Specifying the Multigrid Cycle Type* (p. 1227)). Note that for the parallel solver, efficiency may deteriorate with a W cycle.

If you are using the parallel solver, you can try increasing the maximum number of cycles by increasing the value of **Max Cycles** in the **Multigrid** tab, under **Fixed Cycle Parameters**.

In the pressure-based coupled algorithm and the density-based implicit formulation, there is no pressure correction. Instead, there is a flow correction, which by default uses the F cycle. The density-based explicit formulation uses the V cycle as the default flow correction.

Solution Controls → **Advanced...**

For additional information, please see the following sections:

28.5.1. Specifying the Multigrid Cycle Type
28.5.2. Setting the Termination and Residual Reduction Parameters
28.5.3. Setting the AMG Method and the Stabilization Parameters
28.5.4. Additional Algebraic Multigrid Parameters
28.5.5. Setting FAS Multigrid Parameters

28.5.1. Specifying the Multigrid Cycle Type

By default, the V cycle is used for the pressure equation in the pressure-based segregated algorithm and the flexible cycle is used for all other equations. In the pressure-based coupled algorithm and the density-based implicit formulation, the F cycle is default for the flow correction. The V cycle is default for the flow correction in the density-based explicit formulation. (See Multigrid Cycles in the Theory Guide for a description of these cycles.) To change the cycle type for an equation, you will use the top portion of the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.6* (p. 1231)).

For each equation, you can choose **Flexible**, **V-Cycle**, **W-Cycle**, or **F-Cycle** in the adjacent drop-down list.

28.5.2. Setting the Termination and Residual Reduction Parameters

When you use the flexible cycle for an equation, you can control the multigrid performance by modifying the **Termination** and/or **Restriction** criteria for that equation at the top of the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.6* (p. 1231)).

Solution Controls → **Advanced...**

The **Restriction** criterion is the residual reduction tolerance, β in Equation 19–119 in the Theory Guide. This parameter dictates when a coarser grid level must be visited (due to insufficient improvement in the solution on the current level). With a larger value of β, coarse levels will be visited less often (and vice versa). The

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Termination criterion, α in Equation 19–120 in the Theory Guide, governs when the solver should return to a finer grid level (i.e., when the residuals have improved sufficiently on the current level).

For the V, W, or F cycle, the **Termination** criterion determines whether or not another cycle should be performed on the finest (original) level. If the current residual on the finest level does not satisfy Equation 19–120 in the Theory Guide, and the maximum number of cycles has not been performed, ANSYS FLUENT will perform another multigrid cycle. (The **Restriction** parameter is not used by the V, W, and F cycles.)

28.5.3. Setting the AMG Method and the Stabilization Parameters

You can use the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.6* (p. 1231)) to choose between two AMG solvers: aggregative or selective. The aggregative AMG (AAMG) is the default solver that was used in previous versions of ANSYS FLUENT. The selective AMG (SAMG) solver is available only for scalar equations, and is not available in parallel ANSYS FLUENT. These two solvers differ in the way the grids are coarsened and in their interpolation method.

The AAMG solver [*100*] (p. 2214) builds coarse levels by grouping fine level cells to make coarse level cells, and uses piecewise constant interpolation. The SAMG solver [*86*] (p. 2213) builds coarse levels by selecting some of the fine level cells for solution on the coarse level, and tries to approximate the use of linear interpolation.

Due to its use of more accurate interpolation, SAMG has a better convergence rate than AAMG but has a more expensive setup phase. For this reason, AAMG is usually faster if you are only converging one order of magnitude, while SAMG is faster if using a tight multigrid convergence tolerance. SAMG is a good choice for multiphase granular flow problems where a tight convergence tolerance on the pressure equation can be used to avoid volume imbalance errors in the volume fraction equations.

SAMG has advantages in solving problems with strongly varying (anisotropic) diffusive coefficients, which occurs in problems with porous media, conduction with anisotropic thermal conductivities, and multiphase problems. In some cases, using SAMG allows up to a 20% reduction in the number of external iterations for unsteady water-air turbulent flow in bubble columns, and allows increasing the VOF under-relaxation factor in phase separators from 0.2 (when used with AAMG) to 1.

in the **Multigrid** tab (*Figure 28.6* (p. 1231)), you can also choose a stabilization method. If desired, you can choose the bi-conjugate gradient stabilized method [*9*] (p. 2209) (**BCGSTAB**) option or recursive projection method [*78*] (p. 2213) (**RPM**) in order to improve the convergence of the linear solver. BCGSTAB can be preconditioned by any of the AMG solvers and provides stabilization for them whereas RPM stabilizes the AAMG solver. In addition, RPM stabilization is mainly used in conjunction with the coupled pressure-based solver and it is enabled only when this model is selected.

ANSYS FLUENT usually builds diagonally dominant matrices for the linear solver. However, this is not always possible. A linear system with highly dominant off-diagonal coefficients may occur during discretization of complex physical models such as multiphase cavitation. Using the **BCGSTAB** or **RPM** option in such cases can be helpful. In addition, the AMG convergence in parallel can be improved using the **BCGSTAB** option with AMG.

28.5.4. Additional Algebraic Multigrid Parameters

There are several additional parameters that control the algebraic multigrid solver, but there will usually be no need to modify them. These additional scalar and coupled parameters are all contained in the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.6* (p. 1231)).

Solution Controls → **Advanced...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

When using the density-based explicit formulation or the pressure-based solver with any of the segregated algorithms, described in Pressure-Velocity Coupling in the Theory Guide and *Choosing the Pressure-Velocity Coupling Method* (p. 1215), only **Scalar Parameters** are set in the **Multigrid** tab. If you use the density-based implicit or the pressure-based coupled algorithm, described in Coupled Algorithm in the Theory Guide, then you can set the **Coupled Parameters**.

28.5.4.1. Fixed Cycle Parameters

For the fixed (V, W, and F) multigrid cycles, you can control the number of pre- and post-relaxations (β_1 and β_3 in Multigrid Cycles in the Theory Guide). **Pre-Sweeps** sets the number of relaxations to perform before moving to a coarser level. **Post-Sweeps** sets the number to be performed after coarser level corrections have been applied. Normally, under **Scalar Parameters**, one post-relaxation is performed and no pre-relaxations are done (i.e., $\beta_3 = 1$ and $\beta_1 = 0$), but in rare cases, you may need to increase the value of β_1 to 1 or 2. Under **Coupled Parameters**, three post-relaxations are performed by default with no pre-relaxations.

Important

- If you are using AMG with V-cycle to solve an energy equation with a solid conduction model presented with anisotropic or very high conductivity coefficient, there is a possibility of divergence with a default post-relaxation sweep of 1. In such cases you should increase the post-relaxation sweep (to say 2) in the AMG section for better convergence when using the pressure-based segregated algorithms.
- It is recommended that you use the Fixed F-cycle for the energy equation when running parallel ANSYS FLUENT.

28.5.4.2. Coarsening Parameters

For all multigrid cycle types, you can control the maximum number of coarse levels (**Max Coarse Levels** under **Scalar** or **Coupled Parameters**) that will be built by the multigrid solver.

Sets of coarser simultaneous equations are built until the maximum number of levels has been created, or the coarsest level has only 3 equations. Each level has about half as many unknowns as the previous level, so coarsening until there are only a few cells left will require about as much total coarse-level coefficient storage as was required on the fine mesh. Reducing the maximum coarse levels will reduce the memory requirements, but may require more iterations to achieve a converged solution. Setting **Max Coarse Levels** to 0 turns off the algebraic multigrid solver.

Another coarsening parameter you can control is the increase in coarseness on successive levels. The **Coarsen by** parameter specifies the number of fine grid cells that will be grouped together to create a coarse grid cell. The algorithm groups each cell with its closest neighbor, then groups the cell and its closest neighbor with the neighbor's closest neighbor, continuing until the desired coarsening is achieved. Typical values for the scalar parameters are in the range from 2 to 10, with the default value of 2 for the Gauss-Seidel smoother giving the best performance, but also the greatest memory use. For coupled parameters, a default value of 4 (for 2D) and 8 (for 3D) for the ILU smoother give the best performance. You should not adjust this parameter unless you need to reduce the memory required to run a problem.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Depending on the smoother type, Gauss-Seidel or ILU, the **Coarsen by** and **Post-Sweeps** settings should be changed as follows when selecting the non-default smoother type:

ILU
: **Post-Sweeps**=3 and **Coarsen by** = 8

Gauss-Seidel
: **Post-Sweeps**=1 and **Coarsen by** = 2

28.5.4.3. Smoother Types

Two smoother types are available for scalar and coupled parameters. **Gauss-Seidel** is the simplest smoother type and is recommended when using the pressure-based segregated algorithm. **ILU** is more CPU intensive, but has better smoothing properties for block-coupled systems such as the pressure-based coupled solver and the density-based implicit formulation. In other words, the default scalar **Smoother Type** is **Gauss-Seidel**, while the coupled **Smoother Type** is **ILU**. For more information about the two smoother types, see The Coupled and Scalar AMG Solvers in the Theory Guide.

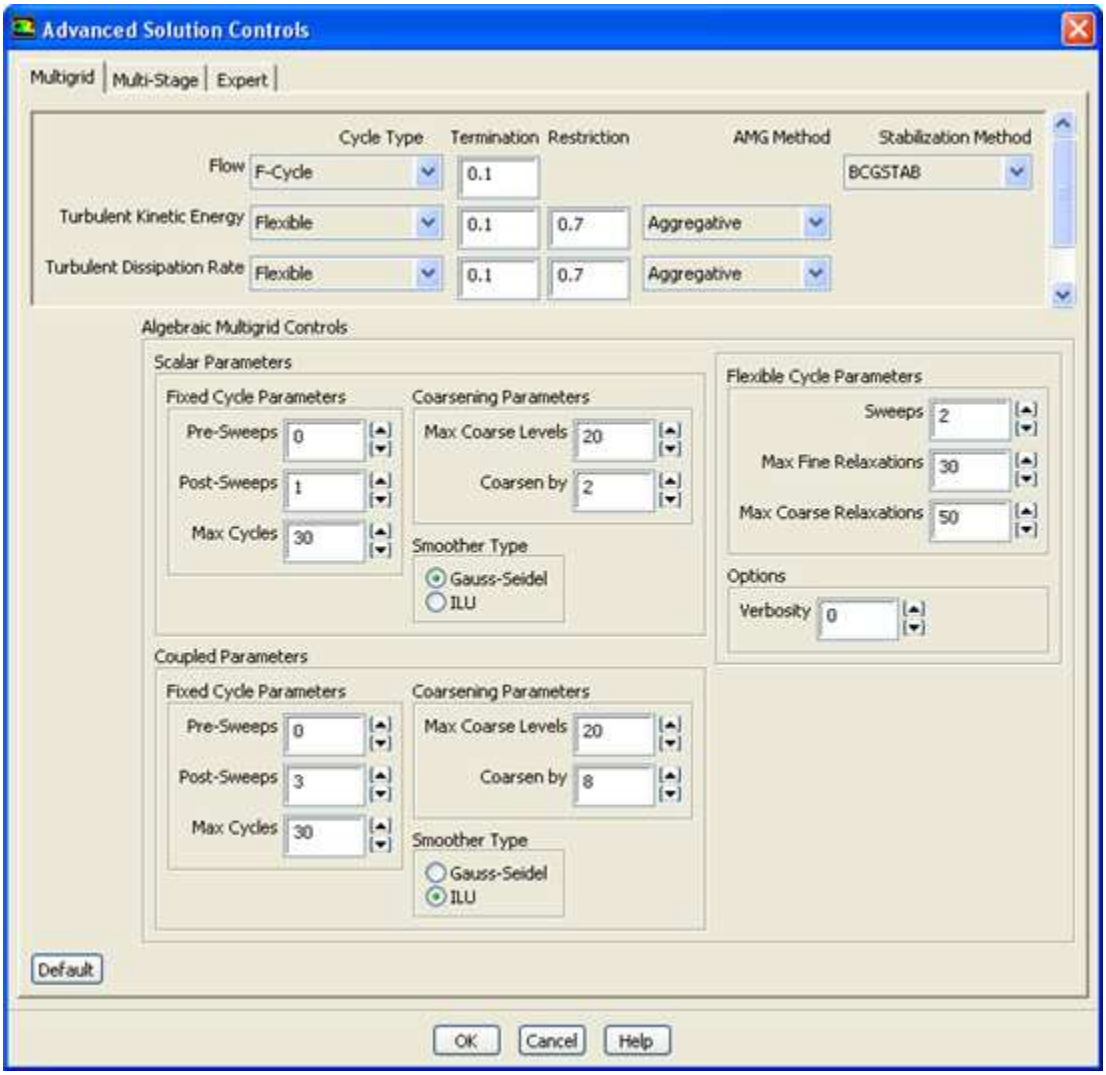
28.5.4.4. Flexible Cycle Parameters

To change the maximum number of relaxations, increase or decrease the value of **Max Fine Relaxations** or **Max Coarse Relaxations** in the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.6* (p. 1231)) under **Flexible Cycle Parameters**.

Solution Controls → **Advanced...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.6 The Multigrid Tab for the Flexible Cycle



28.5.4.5. Setting the Verbosity

The steps for monitoring the solver are as follows:

1. Set multigrid **Verbosity** to 1 or 2 in the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906).

   Solution Controls → **Advanced...**

2. Request a single iteration using the *Run Calculation Task Page* (p. 1955).

   Run Calculation

If you set the verbosity to 2, the information printed in the ANSYS FLUENT console for each equation will include the following:

- equation name
- equation tolerance (computed by the solver using a normalization of the source vector)
- residual value after each fixed multigrid cycle or fine relaxation for the flexible cycle

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- number of equations in each multigrid level, with the zeroth level being the original (finest-level) system of equations

Note that the residual printed at cycle or relaxation 0 is the initial residual before any multigrid cycles are performed.

When verbosity is set to 1, only the equation name, tolerance, and residuals are printed.

A portion of a sample printout is shown below:

```
pressure correction equation:
tol. 1.2668e-05
 0 2.5336e+00
 1 4.9778e-01
 2 2.5863e-01
 3 1.9387e-01

 multigrid levels:
 0 918
 1 426
 2 205
 3  97
 4  45
 5  21
 6  10
 7  4
```

28.5.4.6. Returning to the Default Multigrid Parameters

If you change the multigrid parameters, but you then want to return to ANSYS FLUENT's default settings, you can click the **Default** button in the **Multigrid** tab. ANSYS FLUENT will change all settings to the defaults, and the **Default** button will become the **Reset** button. To get your settings back again, you can click the **Reset** button.

28.5.5. Setting FAS Multigrid Parameters

For most calculations, you will not need to modify any FAS multigrid parameters once you have set the number of coarse grid levels. If, however, you encounter convergence difficulties, you may consider the following suggested procedures.

Important

Recall that FAS multigrid is used only by the density-based explicit formulation.

28.5.5.1. Combating Convergence Trouble

Some problems may approach convergence steadily at first, but then the residuals will level off and the solution will "get stuck." In some cases (e.g., long thin ducts), this convergence trouble may be due to multigrid's slow propagation of pressure information through the domain. In such cases, you should turn off multigrid by setting **Multigrid Levels** to 0 in the *Solution Controls Task Page* (p. 1902).

28.5.5.2. "Industrial-Strength" FAS Multigrid

In some cases, you may find that your problem is converging, but at an extremely slow rate. Such problems can often benefit from a more aggressive form of multigrid, which will speed up the propagation of the solution corrections. For such problems, you can try the "industrial-strength" multigrid settings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

These settings are very aggressive and assume that the solution information passed through the multigrid levels is somewhat accurate. For this reason, you should only attempt the procedure described here after you have performed enough iterations that the solution is off to a good start. Using "industrial-strength" multigrid too early in the calculation process—when the solution is far from correct—will not help convergence and may cause the calculation to become unstable, as very incorrect values are propagated quickly to the original grid. Note also that while these multigrid settings will usually reduce the total number of iterations required to reach convergence, they will greatly increase the computation time for each multigrid cycle. Thus the solver will be performing fewer but longer iterations.

The strategy employed is as follows:

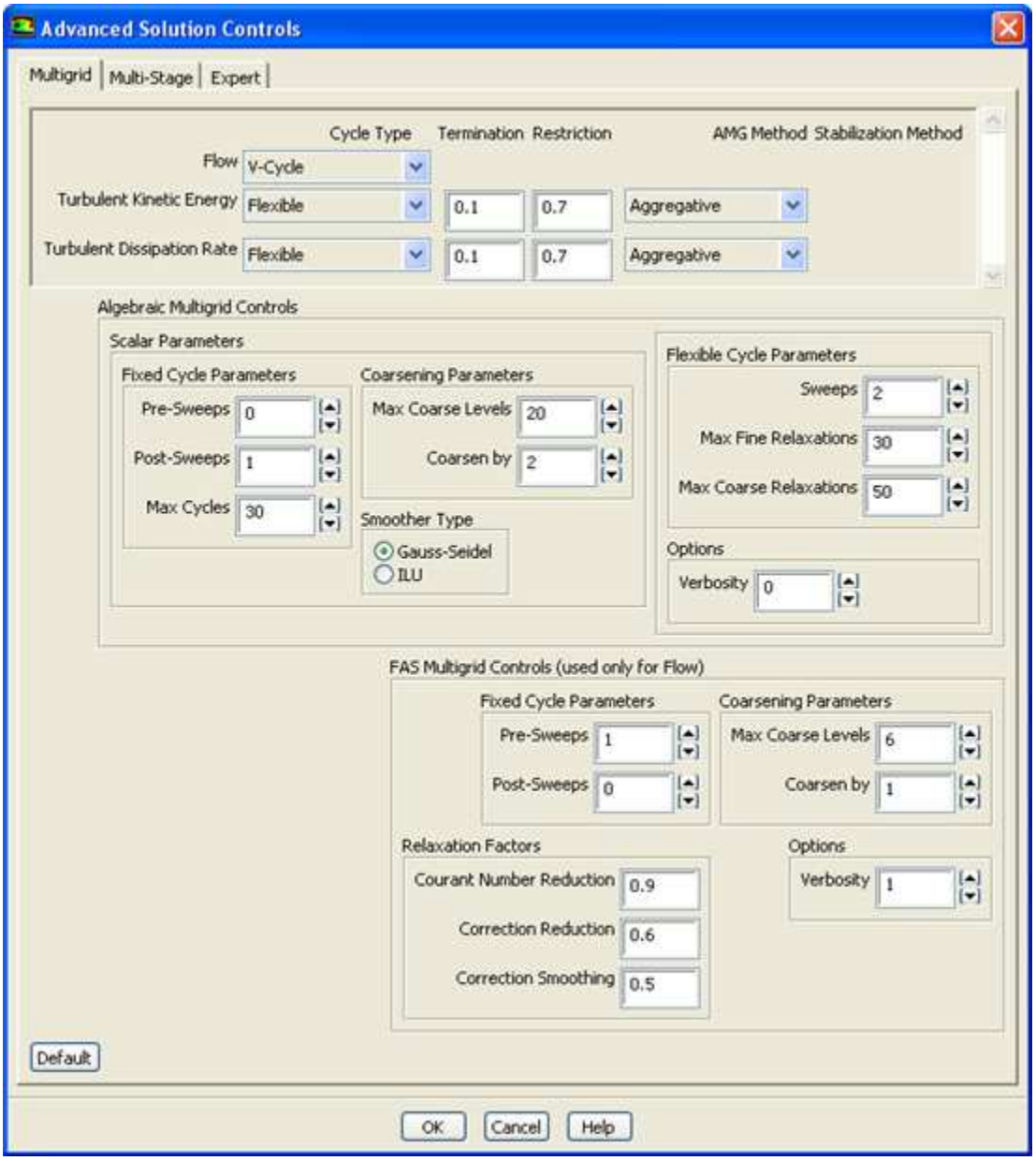
- Increase the number of iterations performed on each grid level before proceeding to a coarser level
- Increase the number of iterations performed on each grid level after returning from a coarser level
- Allow full correction transfer from one level to the next finer level, instead of transferring reduced values of the corrections
- Do not smooth the interpolated corrections when they are transferred from a coarser grid to a finer grid

You can set all of the parameters for this strategy under **FAS Multigrid Controls** in the **Multigrid** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.7* (p. 1234)) and then continue the calculation.

Solution Controls → **Advanced...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.7 The Advanced Solution Controls Dialog Box



Increasing the number of iterations performed on each grid level before proceeding to a coarser level (the value of β_1 described in Multigrid Cycles in the Theory Guide) will improve the solution passed from each finer grid level to the next coarser grid level. Try increasing the value of **Pre-Sweeps** (under **FAS Multigrid Controls**, *not* under **Algebraic Multigrid Controls**) to 10.

Increasing the number of iterations performed on each level after returning from a coarser level will improve the corrections passed from each coarser grid level to the next finer grid level. Errors introduced on the coarser grid levels can therefore be reduced before they are passed further up the grid hierarchy to the original grid. Try increasing the value of **Post-Sweeps** (under **FAS Multigrid Controls**, *not* under **Algebraic Multigrid Controls**) to 10.

By default, the full values of the multigrid corrections are not transferred from a coarser grid to a finer grid; only 60% of the value is transferred. This prevents large errors from transferring quickly up to the original grid and causing the calculation to become unstable. It also prevents a "good" solution from propagating

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

quickly to the original grid. However, by increasing the **Correction Reduction** to 1, you can transfer the full values from coarser to finer grid levels, speeding the propagation of the solution and, usually, the convergence as well. The **Species Correction Reduction** sets the factor by which to reduce the magnitude of the species corrections to stabilize the multigrid calculation. This item appears only when species transport is being modeled.

When the corrections on a coarse grid are passed back to the next finer grid level, the values are, by default, interpolated and then smoothed. Disabling the smoothing so that the actual value in a coarse grid cell is assigned to the fine grid cells that comprise it can also aid convergence. To disable smoothing, set the **Correction Smoothing** to 0. Large discontinuities between cells will be smoothed out implicitly as a result of the additional **Post-Sweeps** performed.

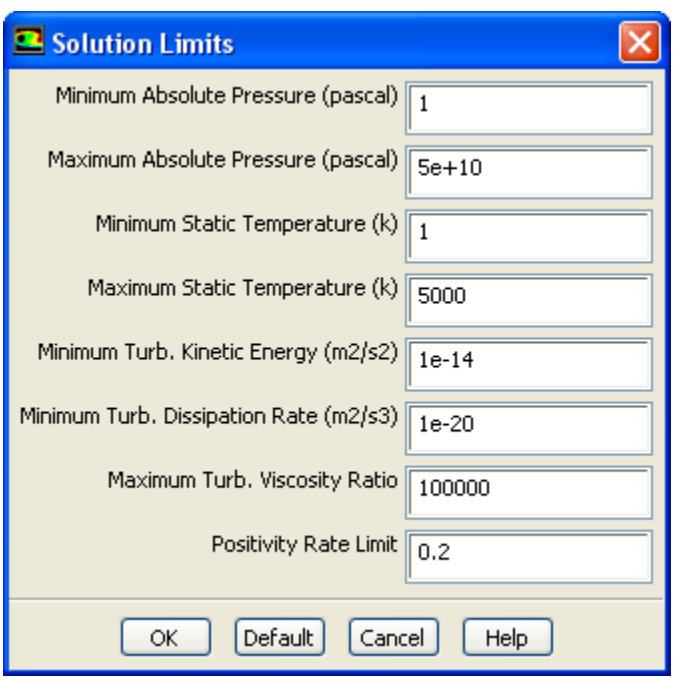
The **Courant Number Reduction** sets the factor by which to reduce the Courant number for coarse grid levels (i.e., every level except the finest). Some reduction of time step (such as the default 0.9) is typically required because the stability limit cannot be determined as precisely on the irregularly shaped coarser grid cells.

28.6. Setting Solution Limits

In order to keep the solution stable under extreme conditions, ANSYS FLUENT provides limits that keep the solution within an acceptable range. You can control these limits with the *Solution Limits Dialog Box* (p. 1904) (*Figure 28.8* (p. 1235)).

Solution Controls → Limits...

Figure 28.8 The Solution Limits Dialog Box



ANSYS FLUENT applies limiting values for pressure, static temperature, and turbulence quantities. The purpose of these limits is to keep the absolute pressure or the static temperature from becoming 0, negative, or excessively large during the calculation, and to keep the turbulence quantities from becoming excessive. ANSYS FLUENT also puts a limit on the rate of reduction of static temperature to prevent it from becoming 0 or negative.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Typically, you will not need to change the default solution limits. If pressure, temperature, or turbulence quantities are being reset to the limiting value repeatedly (as indicated by the appropriate warning messages in the console), you should check the dimensions, boundary conditions, and properties to be sure that the problem is set up correctly and try to determine why the variable in question is getting so close to zero or so large. You can use the "marking" feature (used to mark cells for adaption) to identify which cells have a value equal to the limit. (Use the *Iso-Value Adaption Dialog Box* (p. 2121), as described in *Isovalue Adaption* (p. 1335).) In very rare cases, you may need to change the solution limits, but only do so if you are sure that you understand the reason for the solver's unusual behavior. (For example, you may know that the temperature in your domain will exceed 5000 K. Be sure that any temperature-dependent properties are appropriately defined for high temperatures if you increase the maximum temperature limit.)

Important

For an ideal gas, the absolute pressure and static temperature solution limits are set as described in this section. However, there are no static temperature and absolute pressure solution limits for incompressible flow.

For additional information, please see the following sections:

28.6.1. Limiting the Values of Solution Variables
28.6.2. Adjusting the Positivity Rate Limit
28.6.3. Resetting Solution Limits

28.6.1. Limiting the Values of Solution Variables

The limiting minimum and maximum values for absolute pressure are shown in the **Minimum** and **Maximum Absolute Pressure** fields. If the ANSYS FLUENT calculation predicts a value less than the **Minimum Absolute Pressure** or greater than the **Maximum Absolute Pressure**, the corresponding limiting value will be used instead. Similarly, the **Minimum** and **Maximum Temperature** are limiting values for energy calculations.

The **Maximum Turb. Viscosity Ratio** and the **Minimum Turb. Kinetic Energy** are limiting values for turbulent calculations. If the calculation predicts a k value less than the **Minimum Turb. Kinetic Energy**, the limiting value will be used instead. For the viscosity ratio limit, ANSYS FLUENT uses the limiting maximum value of turbulent viscosity ($C_\mu k^2/\varepsilon$) in the flow field relative to the laminar viscosity. If the ratio calculated by ANSYS FLUENT exceeds the limiting value, the ratio is set to the limiting value by limiting ε to the necessary value.

28.6.2. Adjusting the Positivity Rate Limit

In ANSYS FLUENT's density-based solver, the rate of reduction of temperature is controlled by the **Positivity Rate Limit**. The default value of 0.2, for example, means that temperature is not allowed to decrease by more than 20% of its previous value from one iteration to the next. If the temperature change exceeds this limit, the time step in that cell is reduced to bring the change back into range and a "time step reduced" warning is printed. (This reduced time step will be used for the solution of all variables in the cell, not just for temperature.) Rapid reduction of temperature is an indication that the temperature may become negative. Repeated "time step reduced" warnings should alert you that something is wrong in your problem setup. (If the warning messages stop appearing, the calculation may have "recovered" from the time-step reduction.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

For high-speed flow, if your solution is diverging particularly for the energy equation, then lowering this limit to 0.05 or 0.02 might help in overcoming divergence.

28.6.3. Resetting Solution Limits

If you change and save the value of one of the solution limits, but you then want to return to the default limits set by ANSYS FLUENT, you can reopen the *Solution Limits Dialog Box* (p. 1904) and click the **Default** button. ANSYS FLUENT will change the values to the defaults and the **Default** button will become the **Reset** button. To get your values back again, you can click the **Reset** button.

28.7. Setting Multi-Stage Time-Stepping Parameters

The most common parameter you will change to control the multi-stage time-stepping scheme is the Courant number. Instructions for modifying the Courant number are presented in *Changing the Courant Number* (p. 1222). The **Multi-Stage** tab is accessible from the **Advanced Solution Controls** dialog box when using the density-based explicit formulation.

For additional information, please see the following section:
28.7.1. Changing the Multi-Stage Scheme

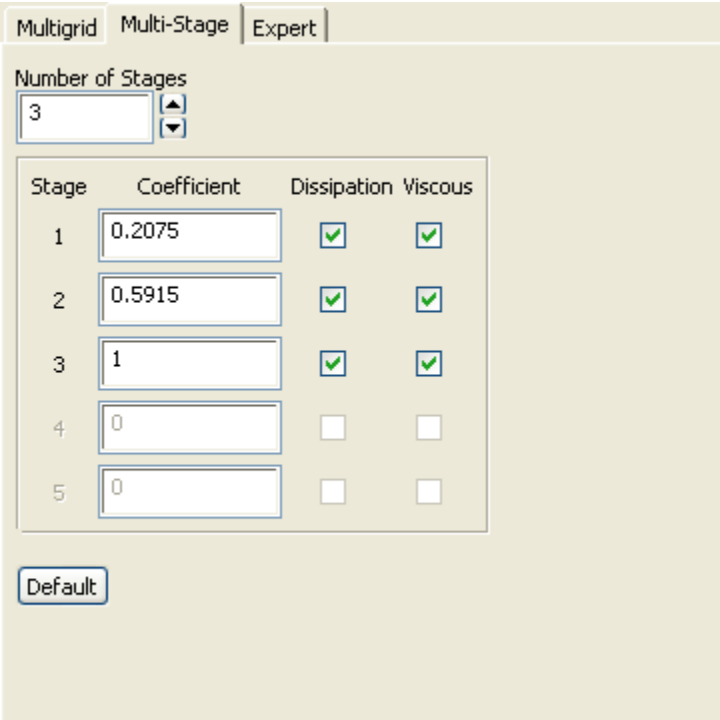
28.7.1. Changing the Multi-Stage Scheme

It is possible to make several changes to the multi-stage time-stepping scheme itself. You can change the number of stages and set a new multi-stage coefficient for each stage. You can also control whether or not dissipation and viscous stresses are updated at each stage. These changes are made in the **Multi-Stage** tab in the *Advanced Solution Controls Dialog Box* (p. 1906) (*Figure 28.9* (p. 1238)).

Solution Controls → **Advanced...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.9 The Multi-Stage Tab



Important

You should not attempt to make changes to ANSYS FLUENT's multi-stage scheme unless you are very familiar with multi-stage schemes and are interested in trying a different scheme found in the literature.

28.7.1.1. Changing the Coefficients and Number of Stages

By default, the ANSYS FLUENT multi-stage scheme uses 3 stages for steady-state solutions with coefficients of 0.2075, 0.5915, and 1.0, and 4 stages for unsteady solutions with coefficients of 0.25, 0.3333, 0.5, and 1.0. You can decrease or increase the number of stages using the arrow buttons for **Number of Stages** in the **Multi-Stage** tab. (If you want to increase the number of stages beyond five, you will need to use the text-interface command `solve/set/multi-stage`.)

For each stage, you can modify the **Coefficient**. Coefficients must be greater than 0 and less than 1. The final stage should always have a coefficient of 1.

28.7.1.2. Controlling Updates to Dissipation and Viscous Stresses

For each stage, you can indicate whether or not artificial dissipation and viscous stresses are evaluated. If a **Dissipation** box is selected for a particular stage, artificial dissipation will be updated on that stage. If not selected, artificial dissipation will remain "frozen" at the value of the previous stage. If a **Viscous** box is selected for a particular stage, viscous stresses will be updated on that stage. If not selected, viscous stresses will remain "frozen" at the value of the previous stage. Viscous stresses should always be computed on the first stage, and successive evaluations will increase the "robustness" of the solution process, but will also increase the expense (i.e., increase the CPU time per iteration). For steady problems, the final solution is independent of the stages on which viscous stresses are updated.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.7.1.3. Resetting the Multi-Stage Parameters

If you change the multi-stage parameters, but you then want to return to the default scheme set by ANSYS FLUENT, you can click the **Default** button in the **Multi-Stage** tab in the *Advanced Solution Controls Dialog Box* (p. 1906). ANSYS FLUENT will change the values to the defaults and the **Default** button will become the **Reset** button. To get your values back again, you can click the **Reset** button.

28.8. Selecting Gradient Limiters

The default gradient limiter in ANSYS FLUENT is the **Standard** limiter. Each of the limiters is described in detail in Gradient Limiters in the Theory Guide. The gradient limiters are accessible from the **Expert** tab in the **Advanced Solution Controls** dialog box.

Solution Controls → **Advanced...**

You can select **Standard**, **Multidimensional**, or **Differentiable** from the **Spatial Discretization Limiter Type** drop-down list.

Each of these options can also be accessed using the TUI by typing the following command:

`solve` → `set` → `slope-limiter-set`

Choose from the following options:

Criterion	Type
0	Default (TVD) slope limiter
1	Multidimensional (TVD) slope limiter
2	Differentiable slope limiter

Note that the `Default(TVD) slope limiter` in the TUI is equivalent to the **Standard** option in the GUI.

For each of the gradient limiter methods, ANSYS FLUENT provides two limiting directions:

- **Cell to Face Limiting** is where the limited value of the reconstruction gradient is determined at cell face centers. This is the default method.
- **Cell to Cell Limiting** is where the limited value of the reconstruction gradient is determined along a scaled line between two adjacent cell centroids. On an orthogonal mesh (or when cell-to-cell direction is parallel to face area direction) this method becomes equivalent to the default cell to face method. For smooth field variation, cell to cell limiting may provide less numerical dissipation on meshes with skewed cells.

28.9. Initializing the Solution

Before starting your CFD simulation, you must provide ANSYS FLUENT with an initial "guess" for the solution flow field. In many cases, you must take extra care to provide an initial solution that will allow the desired final solution to be attained. A real-life supersonic wind tunnel, for example, will not "start" if the back pressure is simply lowered to its operating value; the flow will choke at the tunnel throat and will not transition to supersonic. The same holds true for a numerical simulation: the flow must be initialized to a supersonic flow or it will simply choke and remain subsonic.

There are two methods for initializing the solution:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Initialize the entire flow field (in all cells). Three methods are available:
  - Standard initialization (see *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240))
  - FMG initialization (see *Full Multigrid (FMG) Initialization* (p. 1244))
  - Hybrid initialization (see *Hybrid Initialization* (p. 1245))
- Patch values or functions for selected flow variables in selected cell zones or "registers" of cells. (Registers are created with the same functions that are used to mark cells for adaption.)

Important

Before patching initial values in selected cells, you must first initialize the entire flow field. You can then patch the new values over the initialized values for selected variables.

For additional information, please see the following sections:

28.9.1. Initializing the Entire Flow Field Using Standard Initialization
28.9.2. Patching Values in Selected Cells

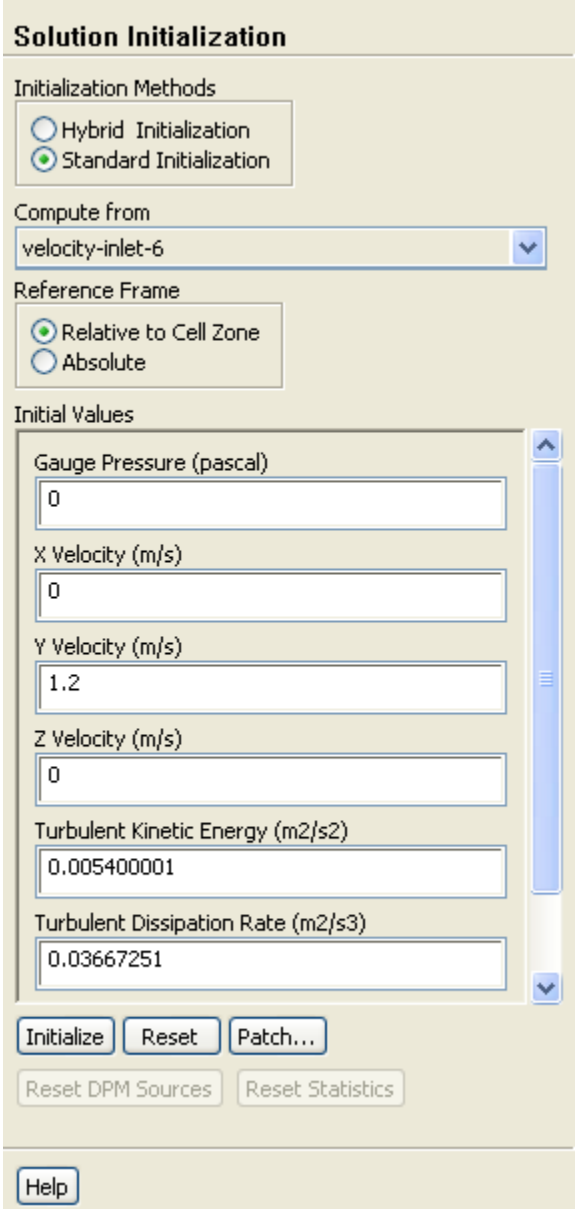
28.9.1. Initializing the Entire Flow Field Using Standard Initialization

Before you start your calculations *or* patch initial values for selected variables in selected cells (*Patching Values in Selected Cells* (p. 1242)) you must initialize the flow field in the entire domain. The *Solution Initialization Task Page* (p. 1937) (*Figure 28.10* (p. 1241)) allows you to set initial values for the flow variables and initialize the solution using these values.

Solution Initialization

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.10 The Solution Initialization Task Page



You can compute the values from information in a specified zone, enter them manually, or have the solver compute average values based on all zones. You can also indicate whether the specified values for velocities are absolute or relative to the velocity in each cell zone. The steps for standard initialization are as follows:

1. Select **Standard Initialization** as the **Initialization Method**.
2. Set the initial values:
   - To initialize the flow field using the values set for a particular zone, select the zone name in the **Compute from** drop-down list. All values under the **Initial Values** heading will automatically be computed and updated based on the conditions defined at the selected zone.
   - To initialize the flow field using computed average values, select **all-zones** in the **Compute from** drop-down list.ANSYS FLUENT will compute and update the **Initial Values** based on the conditions defined at all boundary zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you wish to change one or more of the values, you can enter new values manually in the fields next to the appropriate variables. If you prefer to enter all values manually, you can do so without selecting a zone in the **Compute from** list.

3. If your problem involves moving reference frames or sliding meshes, indicate whether the initial velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone** under **Reference Frame**. (If no zone motion occurs in the problem, the two options are equivalent.) The default reference frame for velocity initialization in ANSYS FLUENT is relative. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.
4. After you are satisfied with the **Initial Values** displayed in the task page, you can click the **Initialize** button to initialize the flow field. If solution data already exist (i.e., if you have already performed some calculations or initialized the solution), you must confirm that it is OK to overwrite those data.

28.9.1.1. Saving and Resetting Initial Values

When you initialize the solution by clicking on **Initialize**, the initial values will also be saved; should you need to reinitialize the solution later, you will find the correct values in the ask page you reopen it.

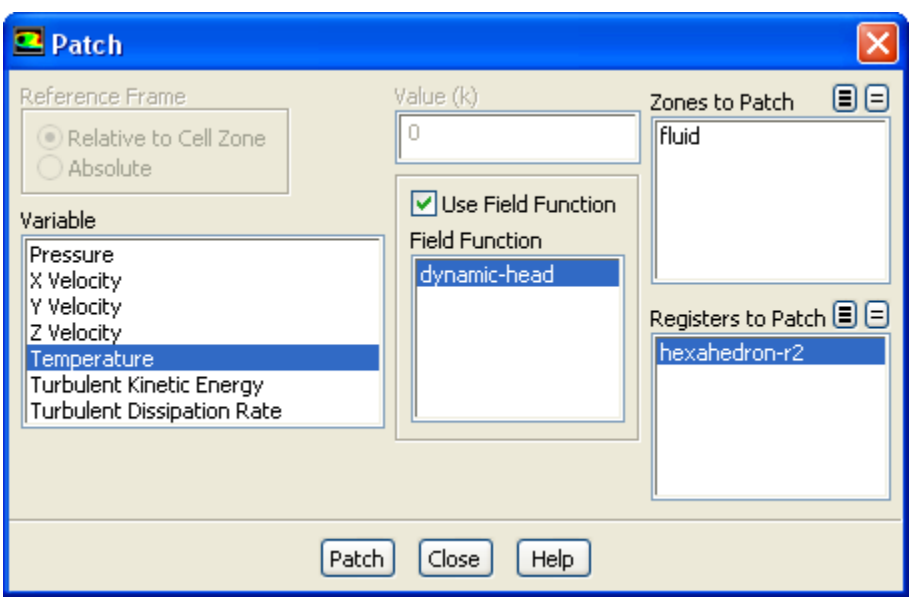
If you accidentally select the wrong zone from the **Compute from** list or manually set a value incorrectly, you can use the **Reset** button to reset all fields to their "saved" values.

28.9.2. Patching Values in Selected Cells

Once you have initialized (or calculated) the entire flow field, you may patch different values for particular variables into different cells. If you have multiple fluid zones, for example, you may want to patch a different temperature in each one. You can also choose to patch a custom field function (defined using the *Custom Field Function Calculator Dialog Box* (p. 2106)) instead of a constant value. If you are patching velocities, you can indicate whether the specified values are absolute velocities or velocities relative to the cell zone's velocity. All patching operations are performed with the *Patch Dialog Box* (p. 1939) (*Figure 28.11* (p. 1242)).

Solution Initialization → **Patch...**

Figure 28.11 The Patch Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the variable to be patched in the **Variable** list.
2. In the **Zones to Patch** and/or **Registers to Patch** lists, choose the zone(s) and/or register(s) for which you want to patch a value for the selected variable.

   Important

   When shell conduction is enabled, the names of the **Zones to Patch**will appear as **shell:wall-name**. The wall-name is the name of the wall on which a shell conduction zone has been created.

3. If you wish to patch a constant value, simply enter that value in the **Value** field. If you want to patch a previously-defined field function, enable the **Use Field Function** option and select the appropriate function in the **Field Function** list.
4. If you selected a velocity in the **Variable** list, and your problem involves moving reference frames or sliding meshes, indicate whether the patched velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone** under **Reference Frame**. (If no zone motion occurs in the problem, the two options are equivalent.) The default reference frame for velocity patching in ANSYS FLUENT is relative. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.
5. Click the **Patch** button to update the flow-field data. (Note that patching will have no effect on the iteration or time-step count.)

28.9.2.1. Using Registers

The ability to patch values in cell registers gives you the flexibility to patch different values within a single cell zone. For example, you may want to patch a certain value for temperature only in fluid cells with a particular range of concentrations for one species. You can create a cell register (basically a list of cells) using the functions that are used to mark cells for adaption. These functions allow you to mark cells based on physical location, cell volume, gradient or isovalue of a particular variable, and other parameters. See *Adapting the Mesh* (p. 1325) for information about marking cells for adaption. *Manipulating Adaption Registers* (p. 1343) provides information about manipulating different registers to create new ones. Once you have created a register, you can patch values in it as described above.

28.9.2.2. Using Field Functions

By defining your own field function using the *Custom Field Function Calculator Dialog Box* (p. 2106), you can patch a non-constant value in selected cells. For example, you may want to patch varying species mass fractions throughout a fluid region. To use this feature, simply create the function as described in *Custom Field Functions* (p. 1579), and then perform the function-patching operation in the *Patch Dialog Box* (p. 1939), as described above.

28.9.2.3. Using Patching Later in the Solution Process

Since patching affects only the variables for which you choose to change the value, leaving the rest of the flow field intact, you can use it later in the solution process without losing calculated data. (Initialization, on the other hand, resets all data to the initial values.) For example, you might want to start a combustion calculation from a cold-flow solution. You can simply read in (or calculate) the cold-flow data, patch a high temperature in the appropriate cells, and continue the calculation.

Patching can also be useful when you are solving a problem using a step-by-step technique, as described in *Step-by-Step Solution Processes* (p. 1318).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.10. Full Multigrid (FMG) Initialization

For many complex flow problems such as those found in rotating machinery, or flows in expanding or spiral ducts, flow convergence can be accelerated if a better initial solution is used at the start of the calculation. The Full Multigrid initialization (FMG initialization) can provide this initial and approximate solution at a minimum cost to the overall computational expense.

For more information about FMG initialization, see Overview of FMG Initialization in the Theory Guide.

For additional information, please see the following sections:

28.10.1. Steps in Using FMG Initialization
28.10.2. Convergence Strategies for FMG Initialization

28.10.1. Steps in Using FMG Initialization

You can access the FMG initialization procedure using the text user interface (TUI) once the standard flow initialization is performed (see Full-Approximation Storage (FAS) Multigrid in the Theory Guide) or if valid flow data is available (i.e., through reading a data file).

To customize the FMG initialization, type the following command :

`solve` → `initialize` → `set-fmg-initialization`

You will be asked to enter:

- The number of multigrid levels for the FMG iteration (the default is 5).

  Important

  For small cases (100, 000 cells or less), it is recommended that you lower the number of multigrid levels to 3 or 4.

- For each level of multigrid, you will be asked to enter the residual reduction (the default value is `0.001`), and the number of cycles per level (the defaults at each level are `10`, `10`, `50`, `100`, `500`, and `500`). In general, you should perform more iterations on coarse levels than fine levels. Level 0 is the finest level, which represents the original mesh.
- FMG iteration Courant-number (the default is `0.75`). This will be the CFL value that the FAS multigrid will use for the FMG initialization.
- Enabling verbose mode (the default is `no`). By enabling this option, you will be able to monitor the convergence at each level.

  Important

  If you do not customize the FMG settings, then the default values will be used.

To perform the FMG initialization, type the following command :

`solve` → `initialize` → `fmg-initialization`

When you are prompted to `Enable FMG initialization? [no]`, type `yes`.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When verbose mode is selected and the FMG initialization is being executed, ANSYS FLUENT will first output the multigrid level information followed by convergence history for the FAS multigrid cycle on each level. The normalized residual value is printed after ten FAS cycles or when the number of FAS cycles is reached. The output will indicate when convergence is reached on each level and when the solution is being interpolated to the next level.

28.10.2. Convergence Strategies for FMG Initialization

When setting the FMG initialization parameters, you should consider performing more iterations on the coarse levels than on the fine levels. However, keep in mind that the purpose of FMG initialization is to obtain a good initial solution at a low cost. You should try to avoid unreasonable convergence tolerance that will make the FMG initialization expensive.

Turn on the verbose mode to help you determine if the flow is converging as expected during the FMG iterations. If the solution is not converging to the desired tolerance, consider increasing the number of FAS multigrid cycles at each level. If the solution is diverging during the FAS cycles, then consider lowering the FMG iteration Courant number since the default value is probably too aggressive and is likely causing the solution to diverge.

For turbulent flows, it is very important to first perform standard initialization with proper and realistic values of the turbulence variables (e.g. k and ε). This can be done by computing the average values based on the conditions defined at the inflow boundary or at all boundary zones. Then, you can proceed with FMG initialization. Unrealistic initialization of turbulence variables may cause convergence difficulties during the first few iterations on the fine mesh, thereby nullifying the benefit of FMG initialization.

28.11. Hybrid Initialization

Hybrid initialization is yet another initialization method in ANSYS FLUENT. The other initialization methods are standard initialization and FMG initialization. Hybrid initialization is a collection of recipes and boundary interpolation methods. It solves Laplace's equation to determine the velocity and pressure fields. All other variables, such as temperature, turbulence, species fractions, volume fractions, etc., will be automatically patched based on domain averaged values or a particular interpolation recipe.

For more information about hybrid initialization, see Hybrid Initialization in the Theory Guide.

For additional information, please see the following sections:

28.11.1. Steps in Using Hybrid Initialization
28.11.2. Solution Strategies for Hybrid Initialization

28.11.1. Steps in Using Hybrid Initialization

To use **Hybrid Initialization**, go to the *Solution Initialization Task Page* (p. 1937) (*Figure 28.12* (p. 1246)) where you will select **Hybrid Initialization**.

Solution Initialization

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

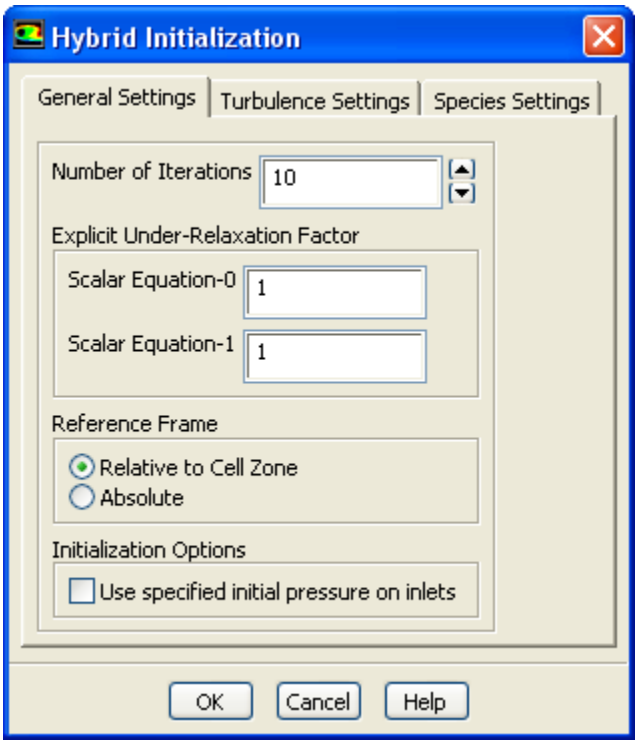
Figure 28.12 The Solution Initialization Task Page for Hybrid Initialization

Note

In most cases, you need not do anything more than click the **Initialize** button. However, should you decide to modify the default settings for the hybrid initialization method, click **More Settings...**.

If you click **More Settings...**, the **Hybrid Initialization** dialog box (*Figure 28.13* (p. 1246)) will open. A host of settings that control the **Hybrid Initialization** strategy will be available for you to adjust.

Figure 28.13 The Hybrid Initialization Dialog Box



You can make adjustments in three different areas:

- **General Settings** tab:
  - **Number of Iterations** uses a default value of 10. This is the number of iterations that will be performed while solving the Laplace equations to initialize the velocity and pressure. In general, you

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

do not need to change the number of iterations. However, for complex and highly curved geometries, if the default number of iterations is not enough to reach the convergence tolerance of 1e-06 and the flow fields are not to your liking, then you may want to increase the number of iterations and re-initialize the flow.

– **Explicit Under-Relaxation Factor** uses a default value of 1. This value will be used while solving the Laplace equation to initialize the velocity and pressure. In general, you do not need to change the explicit under-relaxation factor. However, for some cases, where the scalar residuals are oscillating and showing difficulty reaching the convergence tolerance of 1e-06, you may want to re-initialize the flow by reducing the under-relaxation factor. You may also want to increase the number of iterations to produce a smooth initialization field for the velocity and pressure.

– **Reference Frame** is set to **Relative to Cell Zone** by default. If your problem involves moving reference frames or sliding meshes, indicate whether the initial velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone**. If no zone motion occurs in the problem, the two options are equivalent. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

– **Initialization Option** can be enabled if you want that the specified pressure for **Supersonic/Initialization Gauge Pressure** at the inlet boundaries to be used for solving the Laplace equation for the pressure. Otherwise, ANSYS FLUENT uses a predetermined recipe to determine the initial pressure field, as described in Hybrid Initialization of the Theory Guide.

- **Turbulence Settings** tab uses by default the domain averaged values for the turbulence parameters. If you want to use variable turbulence parameters you can deselect the **Average Turbulent Parameters** check box. When this option is disabled, then it calculates the turbulent parameters, such as kinetic energy, dissipation energy, etc., using local flow parameters.
- **Species Settings** tab will by default initialize secondary species with zero mass or mole fractions. If you want to specify the appropriate value for the species, you will need to enable **Specify Species Parameters**.

28.11.2. Solution Strategies for Hybrid Initialization

In general, you do not need to make any extra adjustments to the hybrid initialization default settings. However, if the hybrid initialization is not producing the initial field to your liking, then you can play with the various options available in the **Hybrid Initialization** dialog box (described in *Steps in Using Hybrid Initialization* (p. 1245)), or you can also use the patching option in addition to the **Hybrid Initialization**. For example, if you are solving a User Defined Scalar, then hybrid initialization will initialize them with a value of zero. However, you can specify the value with which you want to patch.

Note

You can also use a UDF to initialize the flow or certain flow variable in conjunction with hybrid initialization.

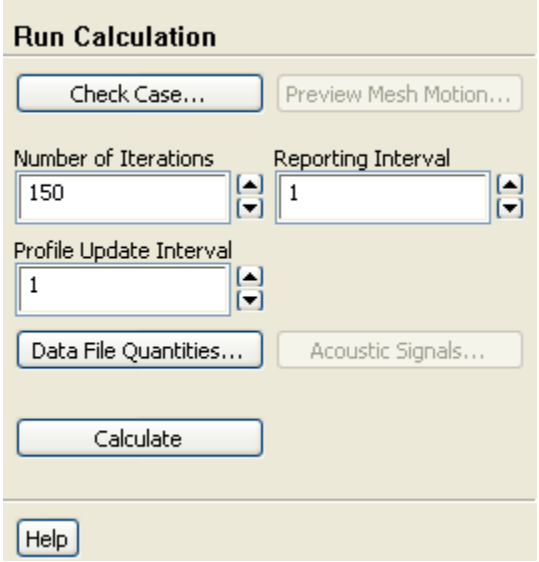
28.12. Performing Steady-State Calculations

For steady-state calculations, you will request the start of the solution process using the *Run Calculation Task Page* (p. 1955) (*Figure 28.14* (p. 1248)).

Run Calculation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.14 The Run Calculation Task Page



Here, you will supply the number of additional iterations to be performed in the **Number of Iterations** field. (For unsteady calculation inputs, see *User Inputs for Time-Dependent Problems* (p. 1256).) If no calculations have been performed yet, ANSYS FLUENT will begin calculations starting at iteration 1, using the initial solution. If you are starting from current solution data, ANSYS FLUENT will begin at the last iteration performed, using the current solution data as its starting point.

By default, ANSYS FLUENT will update the convergence monitors (described in *Monitoring Solution Convergence* (p. 1268)) after each iteration. If you increase the **Reporting Interval** from the default of `1` you can get reports less frequently. For example, if you set the **Reporting Interval** to 2, the monitors will print or plot reports at every other iteration. Note that the **Reporting Interval** also specifies how often ANSYS FLUENT should check if the solution is converged. For example, if your solution converges after 40 iterations, but your **Reporting Interval** is set to 50, ANSYS FLUENT will continue the calculation for an extra 10 iterations before checking for (and finding) convergence.

When you click the **Calculate** button, ANSYS FLUENT will begin to calculate. During iteration, a **Working** dialog box is displayed. Clicking the **Cancel** button or typing `<Control-C>` in the ANSYS FLUENT console will interrupt the iteration, as soon as it is safe to stop. (See below for more details.)

For additional information, please see the following sections:

28.12.1. Updating UDF Profiles
28.12.2. Interrupting Iterations
28.12.3. Resetting Data

28.12.1. Updating UDF Profiles

If you have used a user-defined function (UDF) to define any boundary conditions, properties, etc., you can control the frequency with which the function is updated by modifying the value of the **UDF Profile Update Interval**. If **UDF Profile Update Interval** is set to n, the function will be updated after every n iterations.

By default, the **UDF Profile Update Interval** is set to 1. You might want to increase this value if your profile computation is expensive. See the UDF Manual for details about creating and using UDFs.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.12.2. Interrupting Iterations

As mentioned above, you can interrupt the calculation by clicking the **Cancel** button in the **Working** dialog box that appears while the solver is calculating. In addition, on most, but not all, computer systems you will be able to interrupt calculations using a control sequence, usually `<Control-C>`. This allows you to stop the calculation process before proceeding with the remainder of the requested iterations.

28.12.3. Resetting Data

After you have performed some iterations, if you decide to start over again from the first iteration (e.g., after making some changes to the problem setup), you can reinitialize the solution using the *Solution Initialization Task Page* (p. 1937), as described in *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240).

28.13. Performing Pseudo Transient Calculations

For steady-state calculations, when using the pressure-based coupled solver or the density-based implicit solver, you have the option of solving your flow in a pseudo-transient fashion. The pseudo transient under-relaxation method is a form of implicit under-relaxation, described in Pseudo Transient Under-Relaxation in the Theory Guide.

To apply the pseudo transient under-relaxation method, perform the following:

1. Select the **Density-Based** solver or the **Pressure-Based** solver.

   General

2. Go to the **Solution Methods** task page (*Figure 28.15* (p. 1250)).

   Solution Methods

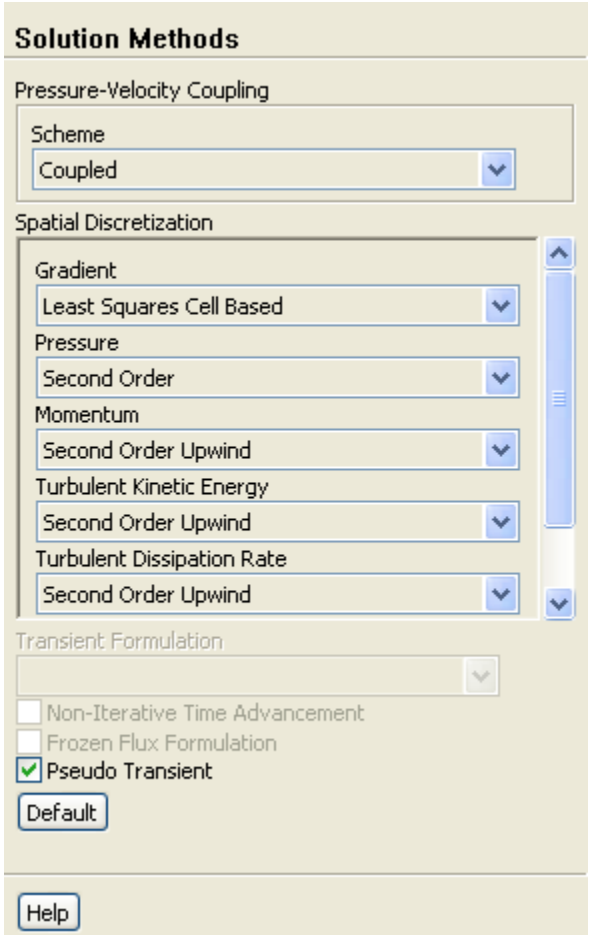
   a. If you are using the pressure-based solver, choose the **Coupled** scheme under **Pressure-Velocity Coupling**.

   b. If you are using the density-based solver, choose the **Implicit** scheme under **Formulation**.

   c. Enable the **Pseudo Transient** option.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.15 The Solution Methods Task Page



28.13.1. Setting Pseudo Transient Explicit Relaxation Factors

In addition to pseudo transient under-relaxation, you can specify an explicit under-relaxation of the equation to control the update of computed variables at each iteration (see Pseudo Transient Under-Relaxation in the Theory Guide). The default values of under-relaxation parameters for all variables are set to values that work well for most of the cases. It is good practice to start a calculation with the default under-relaxation parameters. If your case exhibits divergence or the residuals continue to increase after a few iterations, then you should reduce the under-relaxation factors.

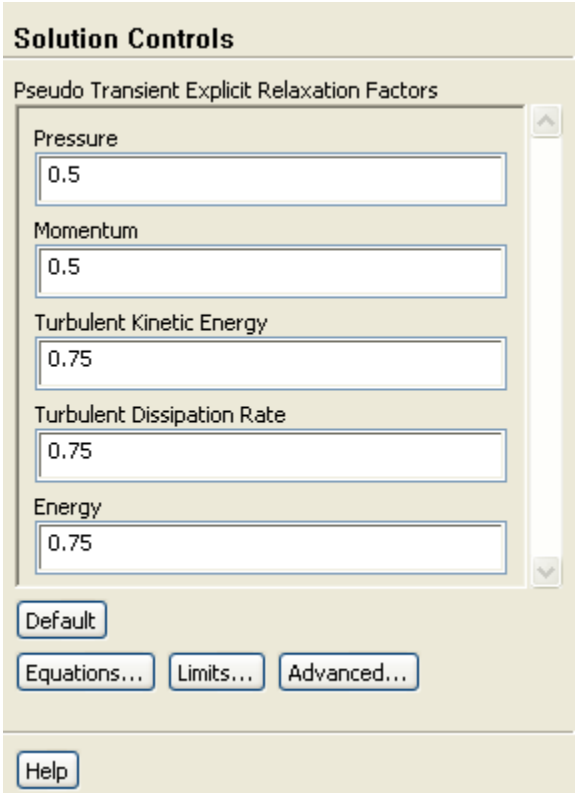
28.13.1.1. User Inputs

You can modify the pseudo transient under-relaxation factors in the *Solution Controls Task Page* (p. 1902) (*Figure 28.16* (p. 1251)).

Solution Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.16 The Solution Controls Task Page for the Pseudo Transient Runs



You can set the under-relaxation factor for each equation in the field next to its name under **Pseudo Transient Explicit Relaxation Factors**.

Important

If you are using the pressure-based solver, all equations will have an associated under-relaxation factor (see Under-Relaxation of Equations in the Theory Guide). If you are using the density-based solver, only those equations that are solved sequentially (see Density-Based Solver in the Theory Guide) will have under-relaxation factors.

If you change under-relaxation factors, but you then want to return to ANSYS FLUENT's default settings, you can click the **Default** button.

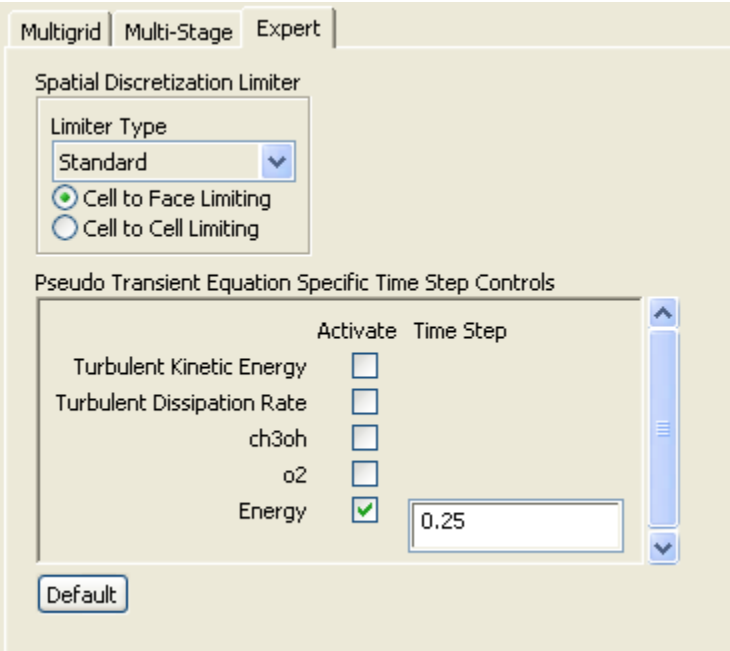
28.13.2. Setting Solution Controls for the Pseudo Transient Method

To have further control over the time steps specified for each individual equation, when solving a pseudo transient case, you can go to the **Expert** tab, in the **Advanced Solution Controls** dialog box (*Figure 28.17* (p. 1252)). Note that all equations, except for flow equations (i.e. pressure and momentum) will be listed. Generally, you will not need to visit this dialog box to enter equation-specific time steps, however it may help in cases where a particular equation is giving convergence problems when using a global pseudo time step.

Solution Controls → Advanced...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.17 The Advanced Solution Controls Dialog Box for the Pseudo Transient Method



Specify the equation-specific time step for a particular equation (if needed) by enabling the **Activate** check box next to the equation and then specifying the pseudo **Time Step** for the equation.

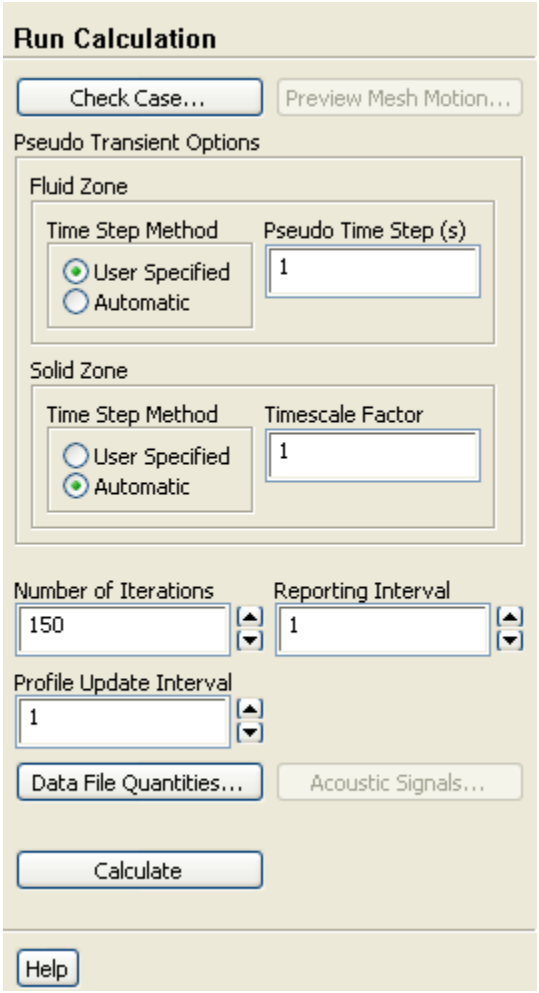
28.13.3. Solving Pseudo-Transient Flow

With the **Pseudo Transient** option enabled in the **Solution Methods** task page, you can now specify the time step for the **Fluid Zone** and/or the **Solid Zone** under **Pseudo Transient Options** in the **Run Calculation** task page (*Figure 28.18* (p. 1253)).

Run Calculation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.18 The Run Calculation Task Page for the User Specified Pseudo Transient Option



1. Select the **Time Step Method** for the **Fluid Zone**.

   a. If you choose **User Specified**, you will enter the **Pseudo Time Step**, which is used for every equation unless the equation specific time step is used for a particular equation, as mentioned in *Setting Solution Controls for the Pseudo Transient Method* (p. 1251).

   b. If you choose **Automatic**, which is the default method, (see *Figure 28.19* (p. 1254)), the pseudo time step is calculated internally and used for each equation listed, unless the equation has a time step specified as noted in *Setting Solution Controls for the Pseudo Transient Method* (p. 1251). For more information about the automatic time step calculation, please refer to Automatic Pseudo Transient Time Step in the Theory Guide.

      Select the Length Scale Method. Three options are available in the drop-down list:

      i. **Conservative** is the default length scale calculation method, for which you will enter the **Timescale Factor**. The **Timescale Factor** is a scaling factor used to scale the calculated time step. The default value is 1.0. You can increase or decrease it to increase or decrease the size of the time step, respectively.

      ii. **Aggressive** predicts a higher time step size than the **Conservative** method. You will also need to specify the **Timescale Factor**.

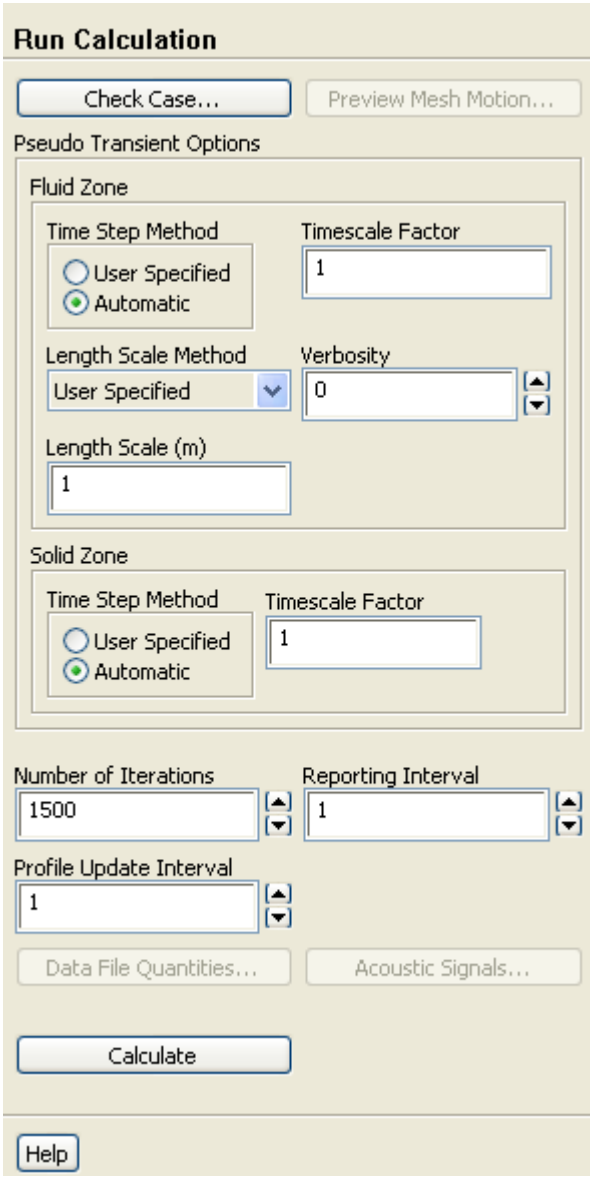
      iii. **User Specified** allows you to control the input for the **Length Scale**. This method is particularly useful in problems which has a specific length scale that is difficult to determine from the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overall geometry of the problem. For example, in the case of flow over an airfoil, the appropriate length scale would be the length of the airfoil instead of the length scale calculated based on the geometry of the entire domain.

iv. You can specify the **Verbosity**. This is an integer value of 0 or 1. The default value is 0. If you want to print the pseudo time step size, enter a value of 1 for the **Verbosity**.

Figure 28.19 The Run Calculation Task Page for the Automatic Pseudo Transient Option



2. Select the **Time Step Method** for the **Solid Zone**.

Note

Solid Zone will only appear in the interface when a solid zone is present in the domain. In solid zones, the same entry fields (e.g. **Timescale Factor**) as the fluid zones take on a different meaning.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. If you choose **User Specified**, you will enter the **Pseudo Time Step**, which is only used for the solid zone.

b. If you choose **Automatic**, which is the default method for solid zones, the pseudo time step is calculated internally as described in Automatic Pseudo Transient Time Step in the Theory Guide. Specify the **Timescale Factor** for the solid zone. You can control the time step in solid zones by adjusting the **Timescale Factor**.

3. Continue setting up your solution as you would a steady-state run, as described in *Performing Steady-State Calculations* (p. 1247).

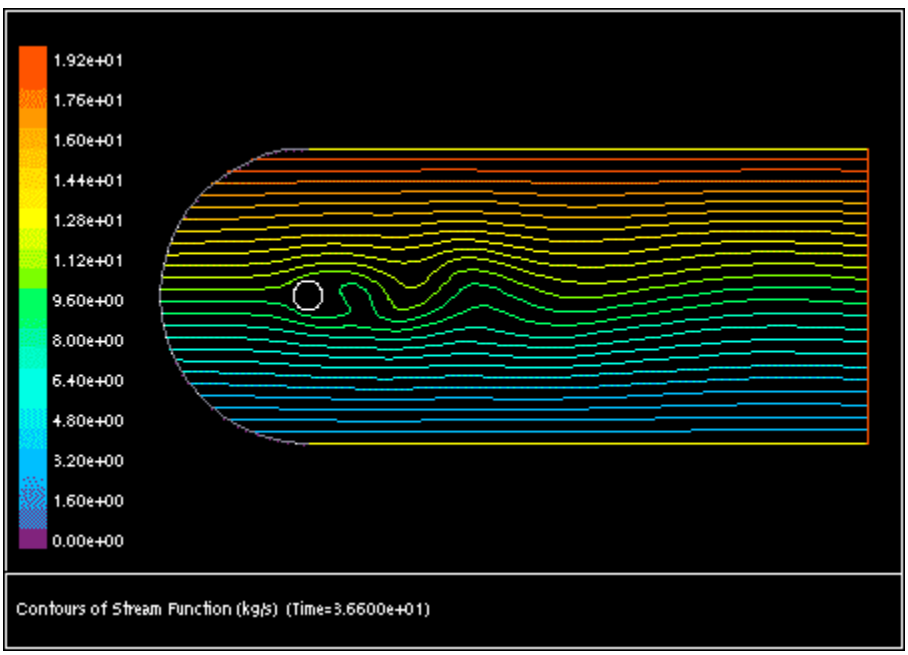
28.14. Performing Time-Dependent Calculations

ANSYS FLUENT can solve the conservation equations in a time-dependent manner, to simulate a wide variety of time-dependent phenomena, such as

- vortex shedding and other time-periodic phenomena
- compressible filling and emptying problems
- transient heat conduction
- transient chemical mixing and reactions

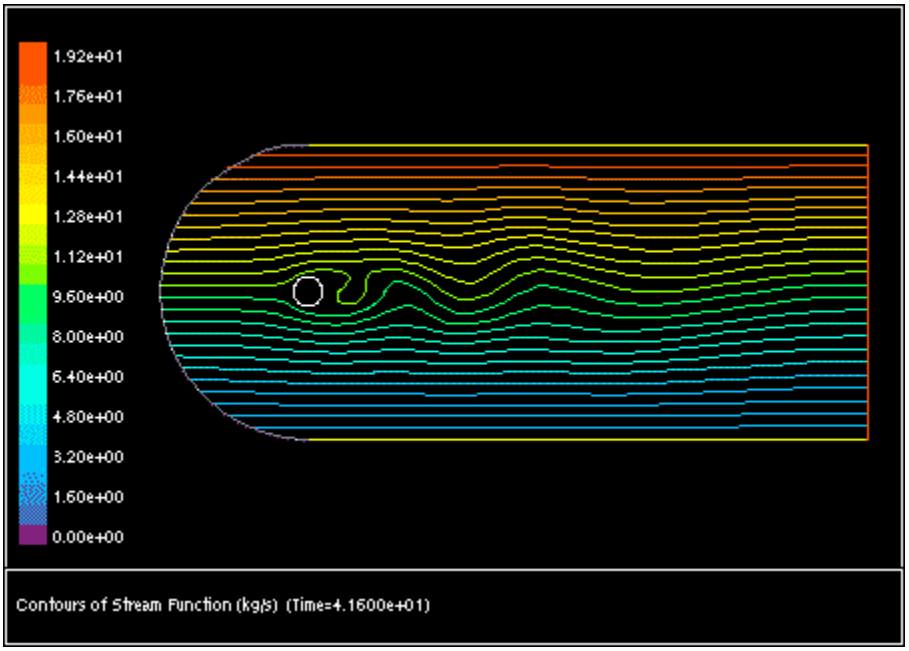
Figure 28.20 (p. 1255) and *Figure 28.21* (p. 1256) illustrate the time-dependent vortex shedding flow pattern in the wake of a cylinder.

Figure 28.20 Time-Dependent Calculation of Vortex Shedding (t=36.6 sec)



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.21 Time-Dependent Calculation of Vortex Shedding (t=41.6 sec)



Activating time dependence is sometimes useful when attempting to solve steady-state problems which tend toward instability (e.g., natural convection problems in which the Rayleigh number is close to the transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.

For details about temporal discretization, see Temporal Discretization in the Theory Guide.

For additional information, please see the following sections:

28.14.1. User Inputs for Time-Dependent Problems
28.14.2. Adaptive Time Stepping
28.14.3. Variable Time Stepping
28.14.4. Postprocessing for Time-Dependent Problems

28.14.1. User Inputs for Time-Dependent Problems

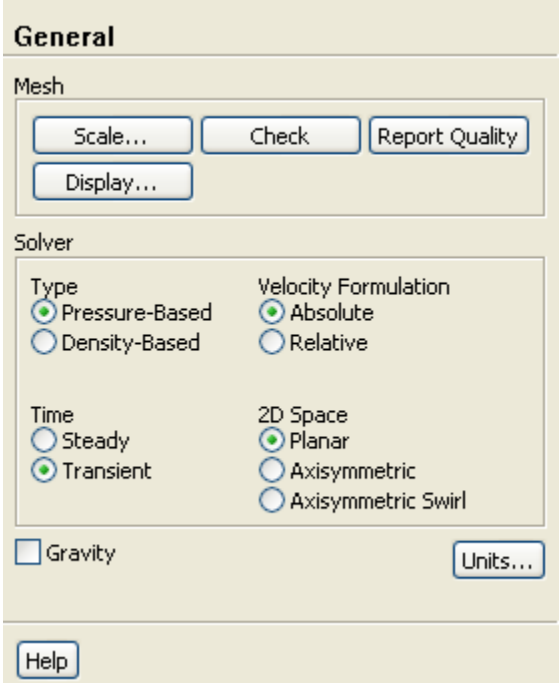
To solve a transient problem, you will follow the procedure outlined below:

1. Enable the **Transient** option in the **General** task page (*Figure 28.22* (p. 1257)).

   General

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

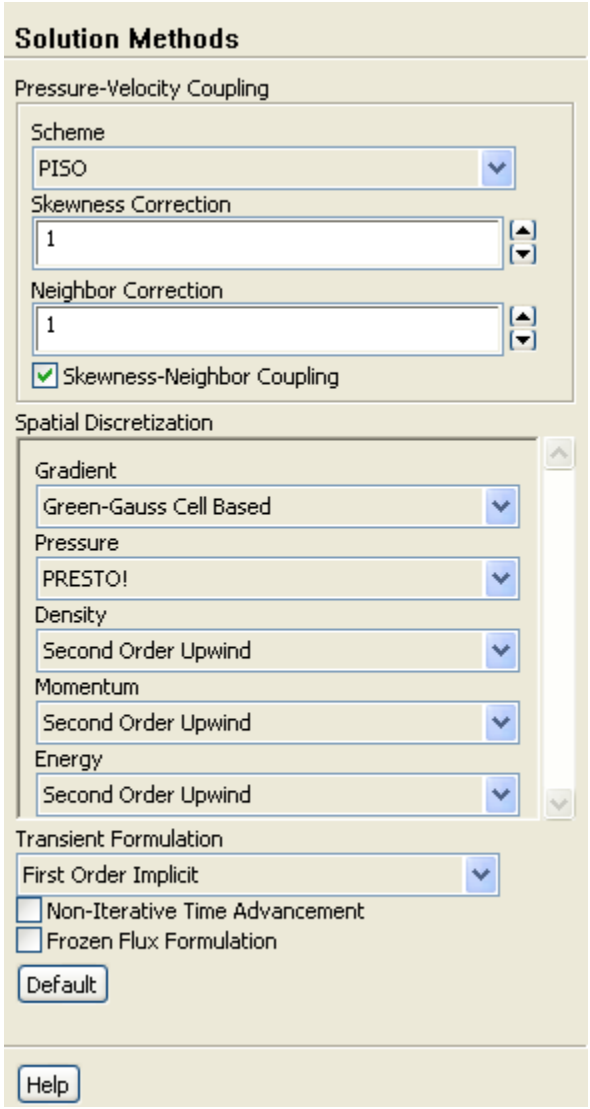
Figure 28.22 The General Task Page for a Transient Calculation



2. Define all relevant models and boundary conditions. Note that any boundary conditions specified using user-defined functions can be made to vary in time. See the UDF Manual for details.
3. Specify the desired parameters in the **Solution Methods** task page (*Figure 28.23* (p. 1258)).

Solution Methods

Figure 28.23 The Solution Methods Task Page for a Transient Calculation



If you are using the pressure-based solver, select **PISO** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box. To increase the speed of the calculations, you may need to modify the parameters related to the PISO scheme from their default values. See *PISO* (p. 1216) for more information about the optimal use of the PISO algorithm.

Important

If you are using the LES turbulence model with small time steps, the PISO scheme may be too computationally expensive. It is therefore recommended that you select **SIMPLE** or **SIMPLEC** instead of **PISO**.

Important

It is best to select the **Coupled** pressure-velocity coupling scheme if you are using large time steps to solve your transient flow, or if you have a poor quality mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Next, specify the desired **Transient Formulation**. The **First Order Implicit** formulation is sufficient for most problems. If you need improved accuracy, you can either use **Second Order Implicit** or **Bounded Second Order Implicit**. The **Bounded Second Order Implicit** formulation would provide better stability, since time discretization would always ensure the bounds for variables, if available.

Important

Note that while the **Bounded Second Order Implicit** formulation provides the same accuracy as the **Second Order Implicit** formulation, it actually provides better stability.

Important

The **Bounded Second Order Implicit** formulation is available only for the pressure-based solver, and not for the density-based solver.

The **Explicit** formulation (available only for the density-based solver) is used primarily to capture the transient behavior of moving waves, such as shocks. For details, see Temporal Discretization in the Theory Guide.

When using the pressure-based solver, you have the additional options of selecting **Non-Iterative Time Advancement** and **Frozen Flux Formulation** for your time-dependent flow calculations (see Time-Advancement Algorithm and Steady-State Iterative Algorithm in the Theory Guide, respectively). Note that the latter option is only available for single-phase transient problems that do not use a moving/deforming mesh model.

4. (optional) If you are using the explicit transient formulation *or* if you are using the adaptive time stepping method (described in a later step and in *Adaptive Time Stepping* (p. 1264)) it is recommended that you enable the printing of the current time (for the explicit transient formulation) or the current time step size (for the adaptive time stepping method) at each iteration, using the *Statistic Monitors Dialog Box* (p. 1917).

   Monitors → **Statistic** → **Edit...**

   Make sure that the desired item is selected from the **Statistics** selection list (**time** for the current time or **delta_time** for the current time step size) and enable the **Print** option. When ANSYS FLUENT prints the residuals to the console window at each iteration, it will include a column with the current time or the current time step size.

5. (optional) Use the *Drag Monitor Dialog Box* (p. 1918), the *Lift Monitor Dialog Box* (p. 1920), the *Moment Monitor Dialog Box* (p. 1921), or the *Surface Monitor Dialog Box* (p. 1923) to monitor (and/or save to a file) time-varying force coefficient values or a report of a field variable or function on a surface as it changes with time. See *Monitoring Solution Convergence* (p. 1268) for details.

6. Set the initial conditions (at time $t=0$) using the **Solution Initialization** task page.

   Solution Initialization

   You can also read in a steady-state data file to set the initial conditions.

   File → **Read** → **Data...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Use the **Autosave** dialog box to specify the file name and frequency with which case and data files should be saved during the solution process. To open the **Autosave** dialog box, click the **Edit...** button next to **Autosave Every** in the **Calculation Activities** task page.

   Calculation Activities (**Autosave Case/Data**) → **Edit...**

   See *Automatic Saving of Case and Data Files* (p. 65) for details about automatic file saving.

   The **Calculation Activities** task page also allows you to export solution and particle history data during the transient calculation. See *Exporting Data During a Transient Calculation* (p. 98) for details.

   If you want to create a graphical animation of the solution over time, you can use the *Solution Animation Dialog Box* (p. 1953) to set up the graphical displays that you want to use in the animation. See *Animating the Solution* (p. 1296) for details.

   You may also want to request automatic execution of other commands using the *Execute Commands Dialog Box* (p. 1950). See *Executing Commands During the Calculation* (p. 1288) for details.

8. (optional) You can improve the convergence of the transient calculations by enabling the **Extrapolate Variables** option in the *Run Calculation Task Page* (p. 1955). This option instructs ANSYS FLUENT to predict the solution variable values for the next time step using a Taylor series expansion, and then inputs that predicted value as an initial guess for the inner iterations of the current time step. As a result, the absolute residual levels are lowered.

   Note that the **Extrapolate Variables** option is not available if you are employing either the NITA scheme with the pressure-based solver or the explicit formulation with the density-based solver.

   Important

   If you use the **Extrapolate Variables** option when modeling an incompressible flow with the density-based solver, it is recommended that you disable the extrapolation of pressure values. After you have enabled the **Extrapolate Variables** option, type the following text command in the console window:

```
> solve/set/extrapolate-eqn-vars/pressure

Extrapolate Pressure? [yes] no
```

9. (optional) If you want ANSYS FLUENT to gather data for time statistics (i.e., time-averaged and root-mean-square values for solution variables) during the calculation, follow these steps:

   a. Enable the **Data Sampling for Time Statistics** option in the *Run Calculation Task Page* (p. 1955) and specify the **Sampling Interval**.

      Run Calculation

      Enabling this option will allow you to display and report both the mean and the root-mean-square (RMS) values, as described in *Postprocessing for Time-Dependent Problems* (p. 1267).

      To select the variables for which you want to collect statistics, click the **Sampling Options...** button.

      Run Calculation → **Sampling Options...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that gathering data for time statistics is not meaningful inside a moving cell zone (e.g., a sliding zone in a sliding mesh problem, a moving zone in a dynamic mesh problem).

b. Initialize the flow statistics.

Solution Initialization → **Reset Statistics**

Note that you can also reset the flow statistics after you have gathered some data for time statistics. If you perform, say, 10 time steps with the **Data Sampling for Time Statistics** option enabled, check the results, and then continue the calculation for 10 more time steps, the time statistics will include the data gathered in the first 10 time steps unless you reinitialize the flow statistics.

10. Specify time-dependent solution parameters and start the calculation as described below for the implicit and explicit transient formulations:

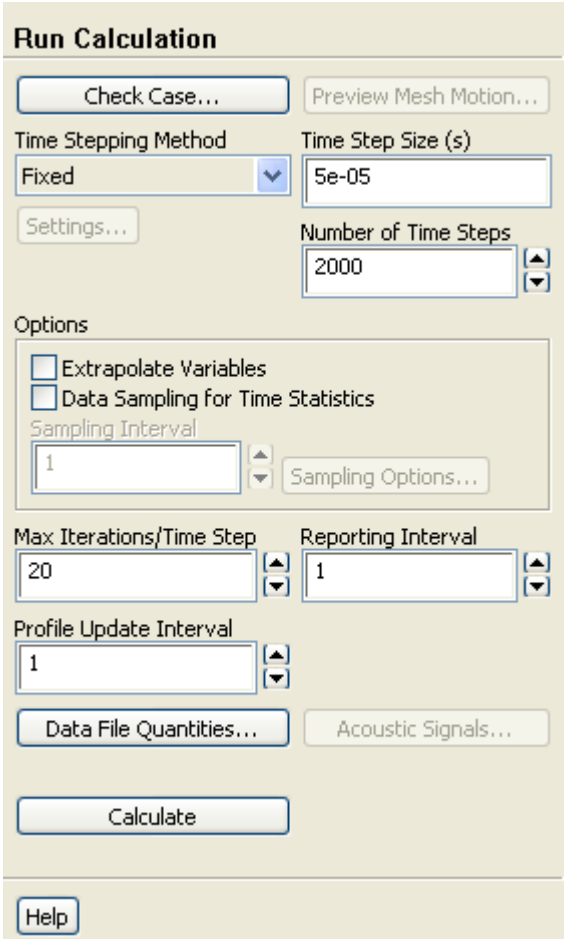
- If you have chosen the **First Order Implicit**, **Second Order Implicit**, or **Second Order Implicit** formulation, the procedure is as follows:

  a. Set the time-dependent solution parameters in the *Run Calculation Task Page* (p. 1955) (see *Figure 28.24* (p. 1262)).

  Run Calculation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.24 The Run Calculation Task Page for Implicit Transient Calculations



Solution parameters for the implicit transient formulations are as follows:

– **Max Iterations/Time Step**: When ANSYS FLUENT solves the time-dependent equations using the implicit formulation, multiple iterations may be necessary at each time step. This parameter sets a maximum for the number of iterations per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step.

– **Time Step Size**: The time step size is the magnitude of Δt. Since the ANSYS FLUENT formulation is fully implicit, there is no stability criterion that needs to be met in determining Δt. However, to model transient phenomena properly, it is necessary to set Δt at least one order of magnitude smaller than the smallest time constant in the system being modeled. A good way to judge the choice of Δt is to observe the number of iterations ANSYS FLUENT needs to converge at each time step. The ideal number of iterations per time step is 5–10. If ANSYS FLUENT needs substantially more, the time step is too large. If ANSYS FLUENT needs only a few iterations per time step, Δt should be increased. Frequently a time-dependent problem has a very fast "startup" transient that decays rapidly. Therefore, it is often wise to choose a conservatively small Δt for the first 5–10 time steps. Δt may then be gradually increased as the calculation proceeds.

  For time-periodic calculations, you should choose the time step based on the time scale of the periodicity. For a rotor/stator model, for example, you might want 20 time steps between each blade passing. For vortex shedding, you might want 20 steps per period.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To verify that your choice for Δt was proper after the calculation is complete, you can plot contours of the Courant number within the domain. To do so, select **Velocity...** and **Cell Courant Number** from the **Contours of** drop-down lists in the **Contours** dialog box. For a stable, efficient calculation, the Courant number should not exceed a value of 20–40 in most sensitive transient regions of the domain.

– **Time Stepping Method**: By default, the size of the time step is fixed (as indicated by the selection of **Fixed**).

To have ANSYS FLUENT modify the size of the time step as the calculation proceeds, select **Adaptive** and click the **Settings...** button to specify the parameters in the **Adaptive Time Step Settings** dialog box. See *Adaptive Time Stepping* (p. 1264) for details.

For transient volume of fluid (VOF) calculations that use the explicit scheme of VOF, you can select the **Variable** time stepping method. The parameters set through the **Parameters...** button are in many ways the same as for the adaptive time stepping method, with the exception of specifying a global Courant number (see *Variable Time Stepping* (p. 1266)).

Note that with the **Adaptive** or **Variable** time stepping method, the value you specify for the **Time Step Size** will be the initial size of the time step. As the calculation proceeds, the **Time Step Size** shown in the **Run Calculation** task page will be the size of the *current* time step.

b. Specify the desired **Number of Time Steps** in the **Run Calculation** task page and click **Calculate**.

As it calculates a solution, ANSYS FLUENT will print the current time at the end of each time step.

- If you have chosen the **Explicit** transient formulation, you will follow a different procedure:

a. Use the default settings in the **Solution Controls** task page.

Solution Controls

If you have modified the parameters, you can click the **Default** button to retrieve the default settings.

b. Specify the desired **Number of Iterations** and click **Calculate**.

Run Calculation

Remember that when the explicit transient formulation is used, each iteration is a time step. When ANSYS FLUENT prints the residuals to the console windows, it will include a column with the current time (if you requested this in step 4, above).

- You can access the information saved in a data file, which includes a standard set of quantities that were computed during the calculation, by clicking the **Data File Quantities...** button. More information about this feature is available in *Setting Data File Quantities* (p. 121).

11. Save the final data file (and case file, if you have modified it) so that you can continue the transient calculation later, if desired.

File → **Write** → **Data...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.14.1.1. Additional Inputs

The procedures for setting the reporting interval, updating UDF profiles, interrupting iterations, and resetting data are the same as those for steady-state calculations. See *Performing Steady-State Calculations* (p. 1247) for details.

Important

If you are using a user-defined function in your time-dependent calculation, note that, in addition to being updated after every n iterations (where n is the value of the **UDF Profile Update Interval**), the function will also be updated at the first iteration of each time step.

28.14.2. Adaptive Time Stepping

As mentioned in *User Inputs for Time-Dependent Problems* (p. 1256), it is possible to have the size of the time step change as the calculation proceeds, rather than specifying a fixed size for the entire calculation. This section provides a brief description of the algorithm that ANSYS FLUENT uses to compute the time step size, as well as an explanation of each of the parameters that you can set to control the adaptive time stepping.

Important

Adaptive time stepping is available only with the pressure-based and density-based implicit formulations; it cannot be used with the density-based explicit formulation. In addition, it cannot be used with the discrete phase model, second-order time integration, Euler-Euler multiphase models (Approaches to Multiphase Modeling in the Theory Guide), or user-defined scalars (*User-Defined Scalar (UDS) Transport Equations* (p. 485)).

28.14.2.1. The Adaptive Time Stepping Algorithm

The automatic determination of the time step size is based on the estimation of the truncation error associated with the time integration scheme. If the truncation error is smaller than a specified tolerance, the size of the time step is increased; if the truncation error is greater, the time step size is decreased.

An estimation of the truncation error can be obtained by using a predictor-corrector type of algorithm [*29*] (p. 2210) in association with the time integration scheme. At each time step, a predicted solution can be obtained using a computationally inexpensive explicit method (forward Euler for the first-order unsteady formulation, Adams-Bashford for the second-order unsteady formulation). This predicted solution is used as an initial condition for the time step, and the correction is computed using the non-linear iterations associated with the implicit (pressure-based or density-based) formulation. The norm of the difference between the predicted and corrected solutions is used as a measure of the truncation error. By comparing the truncation error with the desired level of accuracy (i.e., the truncation error tolerance), ANSYS FLUENT is able to adjust the time step size by increasing it or decreasing it.

In cases where the truncation error remains above the specified tolerance, ANSYS FLUENT will try to meet the tolerance within 5 attempts. If this tolerance is met, then the iteration moves on to the next time step. An explicit scheme is used to predict the solution at each time step, then the explicit prediction is corrected with an implicit scheme. The truncation error, which is a function of the difference between the predicted and corrected solutions at a specific time is used to calculate the next time step. However, if the calculated truncation error is greater than the tolerance limit, we have the option of reverting from the currently performed iteration, which is moving from the nth step to n+1th step, and performing the iteration with a smaller time step. Note that this option is not available for moving deforming meshes, sliding meshes, and

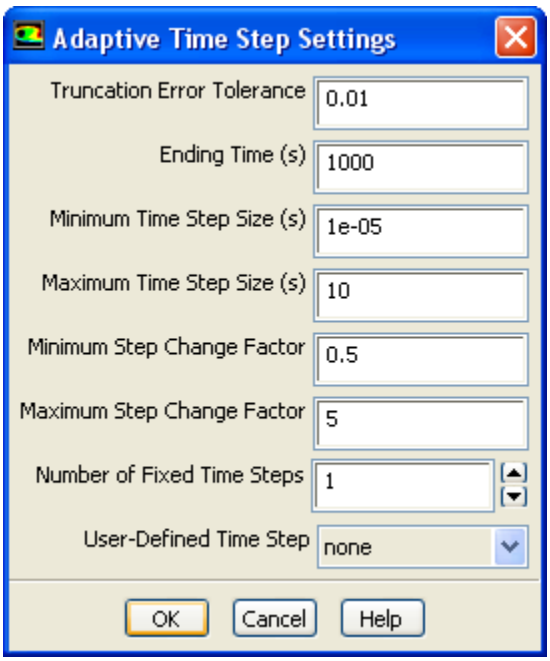
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the discrete phase model. Since the truncation error is proportional to the time step, decreasing the time step reduces the truncation error. This can be done until the truncation error goes below the tolerance limit.

28.14.2.2. Specifying Parameters for Adaptive Time Stepping

The parameters that control the adaptive time stepping appear in the **Adaptive Time Step Settings** dialog box, as described in *User Inputs for Time-Dependent Problems* (p. 1256).

Figure 28.25 The Adaptive Time Step Settings Dialog Box for Implicit Unsteady Calculations and Adaptive Time Stepping



These parameters are as follows:

Truncation Error Tolerance

specifies the threshold value to which the computed truncation error is compared. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

Ending Time

specifies an ending time for the calculation. Since the ending time cannot be determined by multiplying the number of time steps by a fixed time step size, you need to specify it explicitly.

Minimum/Maximum Time Step Size

specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Minimum/Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution. If the time step change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1 < f < f_{max}$, Δt_n is increased to meet the desired tolerance.
- If $1 < f_{max} < f$, Δt_n is increased, but its maximum possible value is $f_{max} \Delta t_{n-1}$.
- If $f_{min} < f < 1$, Δt_n is unchanged.
- If $f < f_{min} < 1$, Δt_n is decreased.

Number of Fixed Time Steps

specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Time Step Size** in the **Run Calculation** task page.

It is a good idea to perform a few fixed-size time steps before switching to the adaptive time stepping. Sometimes spurious discretization errors can be associated with an impulsive start in time. These errors are dissipated during the first few time steps, but they can adversely affect the adaptive time stepping and result in extremely small time steps at the beginning of the calculation.

Important

When the solution tends to exhibit incomplete convergence, rather than increasing the time step size or keeping the same time step size in the next step, ANSYS FLUENT reduces the time step size by at least half for the next time step (making sure that the time step size does not go below the specified minimum time step size.

28.14.2.3. Specifying a User-Defined Time Stepping Method

If you want to use your own adaptive time stepping method, instead of the method described above, you can create a user-defined function for your method and select it in the **User-Defined Time Step** drop-down list. The other inputs in the **Adaptive Time Step Settings** dialog box will not be used when you select a user-defined function.

See the UDF Manual for details about creating and using user-defined functions.

28.14.3. Variable Time Stepping

For VOF and Eulerian multiphase calculations (using the **Explicit** scheme), ANSYS FLUENT allows you to use variable time stepping in order to automatically change the time-step when an interface is moving through dense cells or if the interface velocity is high.

Variable time stepping is available for all the explicit schemes of VOF, which includes the donor-acceptor scheme as well. Variable time stepping is not available for the implicit scheme of VOF.

28.14.3.1. The Variable Time Stepping Algorithm

The global time-step Δt_{global} is changed in the following manner:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

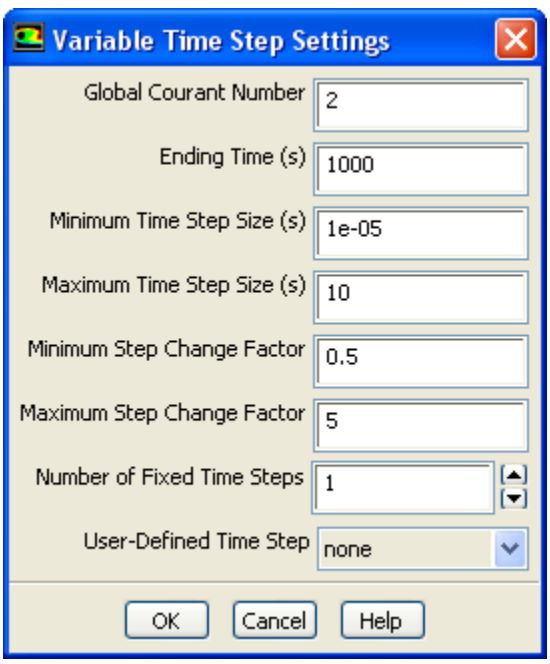
$$\Delta t_{global} = \frac{CFL_{global}}{max\left(\sum \frac{outgoing\ fluxes}{volume}\right)} \tag{28–1}$$

where the ratio $\sum \frac{outgoing\ fluxes}{volume}$ is calculated for each cell. ANSYS FLUENT takes the maximum of this ratio to calculate the global time step.

28.14.3.2. Specifying Parameters for Variable Time Stepping

For transient VOF calculations, when **Variable** is selected from the **Time Stepping Method** drop-down list, in the **Run Calculation** task page and the **Settings...** button is clicked, the **Variable Time Step Settings** dialog box will open (*Figure 28.26* (p. 1267)). With the exception of the **Global Courant Number** field, all parameters are the same as for adaptive time stepping (see *Adaptive Time Stepping* (p. 1264)). The default value for the **Global Courant Number** is 2.

Figure 28.26 The Variable Time Step Settings Dialog Box for Implicit Unsteady Calculations and Variable Time Stepping



The variable time step is based on the maximum Courant number near the VOF interface. To calculate that Courant number, ANSYS FLUENT uses a flux-based definition where, in the region near the fluid interface, ANSYS FLUENT divides the volume of each cell by the sum of the outgoing fluxes. The resulting time represents the time it would take for the fluid to empty out of the cell. The smallest such time is used as the characteristic time of transit for a fluid element across a control volume.

28.14.4. Postprocessing for Time-Dependent Problems

The postprocessing of time-dependent data is similar to that for steady-state data, with all graphical and alphanumeric commands available. You can read a data file that was saved at any point in the calculation (by you or with the autosave option) to restore the data at any of the time levels that were saved.

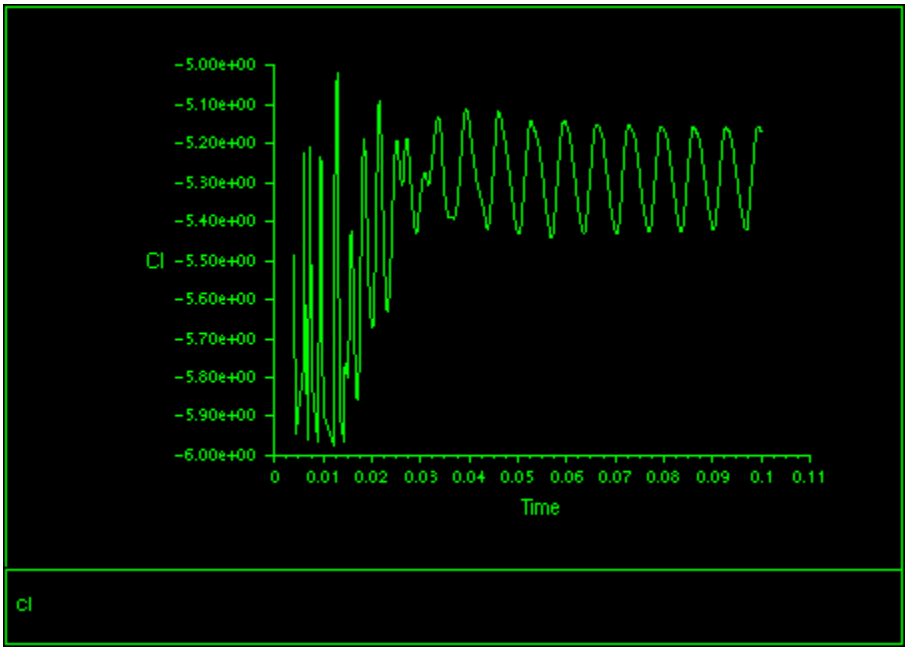
File → **Read** → **Data...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT will label any subsequent graphical or alphanumeric output with the time value of the current data set.

If you save data from the force or surface monitors to files (see step 5 in *User Inputs for Time-Dependent Problems* (p. 1256)), you can read these files back in and plot them to see a time history of the monitored quantity. *Figure 28.27* (p. 1268) shows a sample plot generated in this way.

Figure 28.27 Lift Coefficient Plot for a Time-Periodic Solution



If you enabled the **Data Sampling for Time Statistics** option in the *Run Calculation Task Page* (p. 1955), ANSYS FLUENT will compute the time average (mean) of the instantaneous values and root-mean-squares of the fluctuating values sampled during the calculation. The mean and root-mean-square (RMS) values for all solution variables will be available in the **Unsteady Statistics...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. You can set the **Sampling Interval** such that **Data Sampling for Time Statistics** can be performed at the specified frequency.

28.15. Monitoring Solution Convergence

During the solution process you can monitor the convergence dynamically by checking residuals, statistics, force values, surface integrals, and volume integrals. You can print reports of or display plots of lift, drag, and moment coefficients, surface integrations, and residuals for the solution variables. For unsteady flows, you can also monitor elapsed time. Each of these monitoring features is described below.

For additional information, please see the following sections:

28.15.1. Monitoring Residuals
28.15.2. Monitoring Statistics
28.15.3. Monitoring Force and Moment Coefficients
28.15.4. Monitoring Surface Integrals
28.15.5. Monitoring Volume Integrals

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.15.1. Monitoring Residuals

At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. This history is also saved in the data file. The residual sum is defined below.

On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value ("round-off") and then stop changing ("level out"). For single-precision computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round-off. Double-precision residuals can drop up to twelve orders of magnitude. Guidelines for judging convergence can be found in *Judging Convergence* (p. 1317).

28.15.1.1. Definition of Residuals for the Pressure-Based Solver

After discretization, the conservation equation for a general variable ϕ at a cell P can be written as

$$a_P \phi_P = \sum_{nb} a_{nb} \phi_{nb} + b \qquad \textbf{(28–2)}$$

Here a_P is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells, and b is the contribution of the constant part of the source term S_c in $S = S_c + S_P \phi$ and of the boundary conditions. In *Equation 28–2* (p. 1269),

$$a_P = \sum_{nb} a_{nb} - S_P \qquad \textbf{(28–3)}$$

The residual R^{ϕ} computed by ANSYS FLUENT's pressure-based solver is the imbalance in *Equation 28–2* (p. 1269) summed over all the computational cells P. This is referred to as the "unscaled" residual. It may be written as

$$R^{\phi} = \sum_{cells P} \left| \sum_{nb} a_{nb} \phi_{nb} + b - a_P \phi_P \right| \qquad \textbf{(28–4)}$$

In general, it is difficult to judge convergence by examining the residuals defined by *Equation 28–4* (p. 1269) since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual.

ANSYS FLUENT scales the residual using two kinds of scaling factors, representative of the flow rate of ϕ through the domain. The factors are termed *global scaling* and *local scaling*. The type of scaling can be selected from the **Residual Monitors** dialog box. The "globally scaled" residual is defined as

$$R^{\phi} = \frac{\sum_{cells P} \left| \sum_{nb} a_{nb} \phi_{nb} + b - a_P \phi_P \right|}{\sum_{cells P} \left| a_P \phi_P \right|} \qquad \textbf{(28–5)}$$

For the momentum equations the denominator term $a_P \phi_P$ is replaced by $a_P v_P$, where v_P is the magnitude of the velocity at cell P.

The "locally scaled" residual is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R_{\phi}=\frac{\sqrt{\sum_{cells}^{n}\left(\frac{1}{n}\right)\left(\frac{\Sigma_{nb}a_{nb}\phi_{nb}+b-a_p\phi_p}{a_p}\right)^2}}{\left(\phi_{max}-\phi_{min}\right)_{domain}} \tag{28–6}$$

As in global scaling, for the momentum, ϕ is replaced by v of the cell.

The scaled residual is a more appropriate indicator of convergence for most problems. The selection of scaling and convergence criteria for different types of scaling are discussed in *Judging Convergence* (p. 1317). The default residuals displayed by ANSYS FLUENT are global scaling.

For the continuity equation, the unscaled residual for the pressure-based solver is defined as

$$R^c=\sum_{\text{cells } P}|\text{rate of mass creation in cell P}| \tag{28–7}$$

The local scaling is the same for all equations. However, the global scaling treats continuity in a different way and it is defined as

$$\frac{R^c_{\text{iteration } N}}{R^c_{\text{iteration } 5}} \tag{28–8}$$

The denominator is the largest absolute value of the continuity residual in the first five iterations.

The scaled residuals described above are useful indicators of solution convergence. Guidelines for their use are given in *Judging Convergence* (p. 1317). It is sometimes useful to determine how much a residual has decreased during calculations as an additional measure of convergence. For this purpose, ANSYS FLUENT allows you to normalize the residual (either scaled or unscaled) by dividing by the maximum residual value after M iterations, where M is set by you in the *Residual Monitors Dialog Box* (p. 1915) in the **Iterations** field under **Residual Values**.

$$\overline{R}^{\phi}=\frac{R^{\phi}_{\text{iteration } N}}{R^{\phi}_{\text{iteration } M}} \tag{28–9}$$

Normalization in this manner ensures that the initial residuals for all equations are of $O(1)$ and is sometimes useful in judging overall convergence.

By default, $M=5$. You can also specify the normalization factor (the denominator in *Equation 28–9* (p. 1270)) manually in the *Residual Monitors Dialog Box* (p. 1915).

28.15.1.2. Definition of Residuals for the Density-Based Solver

A residual for the density-based solver is simply the time rate of change of the conserved variable (W). The RMS residual is the square root of the average of the squares of the residuals in each cell of the domain:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R(W) = \sqrt{\sum \left(\frac{\partial W}{\partial t}\right)^2} \tag{28–10}$$

Equation 28–10 (p. 1271) is the unscaled residual sum reported for all the coupled equations solved by ANSYS FLUENT's density-based solver.

Important

The residuals for the equations that are solved sequentially by the density-based solver (turbulence and other scalars, as discussed in Density-Based Solver in the Theory Guide) are the same as those described above for the pressure-based solver.

In general, it is difficult to judge convergence by examining the residuals defined by *Equation 28–10* (p. 1271) since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual. As with the pressure-based solver, ANSYS FLUENT uses two types of scaling for the density-based solver. The globally scaled residual is defined as

$$\frac{R(\mathrm{W})_{\text{iteration } N}}{R(\mathrm{W})_{\text{iteration } 5}} \tag{28–11}$$

The denominator is the largest absolute value of the residual in the first five iterations.

The locally scaled residual is calculated from the local flux imbalance in the cell. It is calculated using *Equation 28–12* (p. 1271):

$$R_\phi = \frac{\sqrt{\sum_n^{cells} \left(\frac{1}{n}\right)\left(\frac{\partial w}{\partial t} V\right)^2}}{(w_{max} - w_{min})_{domain}} \tag{28–12}$$

w is the conservative variable and V is the cell volume. The unscaled residual is always calculated from *Equation 28–10* (p. 1271).

The scaled residuals described above are useful indicators of solution convergence. Guidelines for their use are given in *Judging Convergence* (p. 1317). It is sometimes useful to determine how much a residual has decreased during calculations as an additional measure of convergence. For this purpose, ANSYS FLUENT allows you to normalize the residual (either scaled or unscaled) by dividing by the maximum residual value after M iterations, where M is set by you in the *Residual Monitors Dialog Box* (p. 1915) in the **Iterations** field under **Residual Values**.

Normalization of the residual sum is accomplished by dividing by the maximum residual value after M iterations, where M is set by you in the *Residual Monitors Dialog Box* (p. 1915) in the **Iterations** field under **Residual Values**:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{R}(\mathrm{W}) = \frac{R(\mathrm{W})_{\text{iteration } N}}{R(\mathrm{W})_{\text{iteration } M}} \tag{28–13}$$

Normalization in this manner ensures that the initial residuals for all equations are of $O(1)$ and is sometimes useful in judging overall convergence.

By default, $M = 5$, making the normalized residual equivalent to the scaled residual. You can also specify the normalization factor (the denominator in *Equation 28–13* (p. 1272)) manually in the *Residual Monitors Dialog Box* (p. 1915).

28.15.1.3. Overview of Using the Residual Monitors Dialog Box

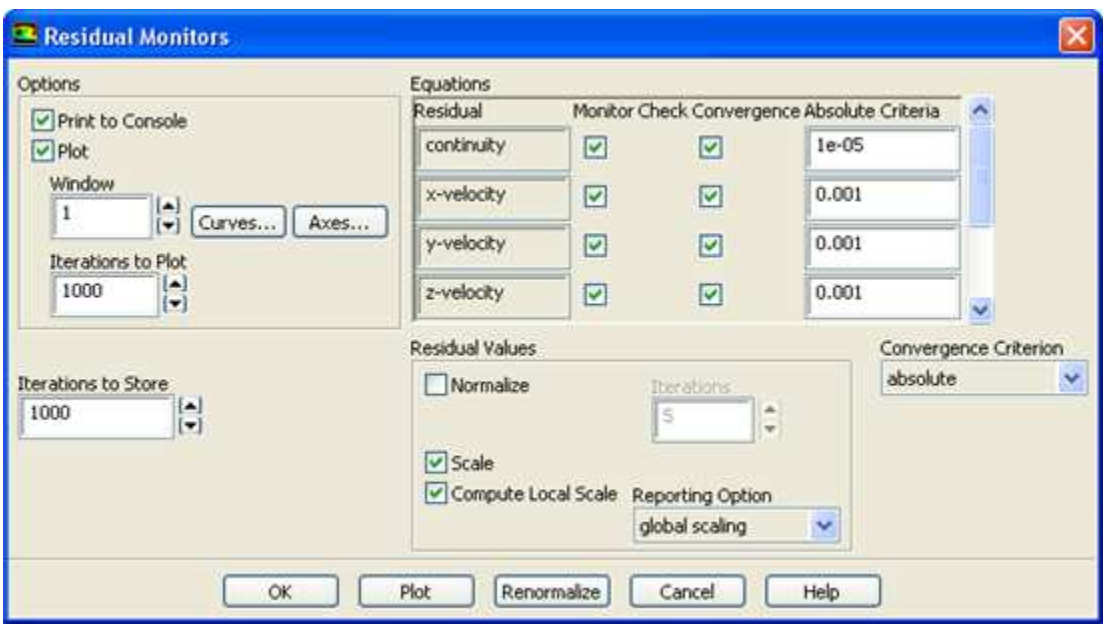
All inputs controlling the monitoring of residuals are entered using the *Residual Monitors Dialog Box* (p. 1915)(*Figure 28.28* (p. 1272)).

Monitors → **Residuals** → **Edit...**

or

Display → **Residuals...**

Figure 28.28 The Residual Monitors Dialog Box



In general, you will only need to enable residual plotting and modify the convergence criteria using this dialog box. Additional controls are available for disabling monitoring of particular residuals, and modifying normalization and plot parameters.

28.15.1.4. Printing and Plotting Residuals

By default, residual values for all relevant variables are printed in the console window after each iteration. If you wish to disable this printout, turn off **Print to Console** under **Options**. To enable the plotting of residuals after each iteration, turn on **Plot** under **Options**. Residuals will be plotted in the graphics window (with the window ID set in the **Window** field) during the calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you wish to display a plot of the current residual history, simply click the **Plot** push button.

28.15.1.5. Storing Residual History Points

Residual histories for each variable are automatically saved in the data file, regardless of whether they are being monitored. You can control the number of history points to be stored by changing the **Iterations to Store** entry. By default, up to 1000 points will be stored. If more than 1000 iterations are performed (i.e., the limit is reached), every other point will be discarded—leaving 500 history points—and the next 500 points will be stored. When the total hits 1000 again, every other point will again be discarded, etc. If you are performing a large number of iterations, you will lose a great deal of residual history information at the beginning of the calculation. In such cases, you should increase the **Iterations to Store** value to a more appropriate value. Of course, the larger this number is, the more memory you will need, the longer the plotting will take, and the more disk space you will need to store the datafile.

28.15.1.6. Controlling Normalization and Scaling

By default, scaling of residuals (see *Equation 28–5* (p. 1269) and *Equation 28–11* (p. 1271)) is enabled and the default convergence criterion is 10^{-6} for energy and P-1 equations and 10^{-3} for all other equations. When the **Scale** option is enabled, global scaling will be applied by default. You can then activate the **Compute Local Scale** option and select the **Reporting Option** from the drop-down list. You have a choice to plot or print to the console the **local scaling** or the **global scaling** of residuals. By default, the **global scaling** of residuals will be plotted.

Note

When **Compute Local Scale** is enabled, ANSYS FLUENT computes and stores both the locally and globally scaled residuals from subsequent iterations, for the purpose of reporting. The scaled residuals are stored in the data file.

Important

Once the **Compute Local Scale** option is activated and you disable the **Scale** option, the **Compute Local Scale** option will not automatically be disabled. In stead, it will compute both the locally and globally scaled residuals, but only print or plot the unscaled residual.

Residual normalization (i.e., dividing the residuals by the largest value during the first few iterations) is also available but disabled by default.

Normalization can be used with both scaled and unscaled residuals. Note that if normalization is enabled, the convergence criterion may need to be adjusted appropriately. See *Judging Convergence* (p. 1317) for information about judging convergence based on the different types of residual reports. (Both the raw residuals and scaling factors are stored in the data file, so you can switch between scaled and unscaled residuals.)

To report unscaled residuals, simply disable the **Scale** option under **Residual Values**.

Important

If you switch from scaled to unscaled residuals (or vice versa) and you are normalizing the residuals (as described below), you must click the **Renormalize** button to recompute the normalization factors.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you wish to normalize the residuals (see *Equation 28–9* (p. 1270) or *Equation 28–13* (p. 1272)), enable the **Normalize** option under **Residual Values**. The **Normalization Factor** column will be added to the dialog box at this time. ANSYS FLUENT will normalize the printed or plotted residual for each variable by the value indicated as the **Normalization Factor** for that variable. The default **Normalization Factor** is the maximum residual value after the first 5 iterations. To use the maximum residual value after a different number of iterations (i.e., specify a different value for M in *Equation 28–9* (p. 1270) or *Equation 28–13* (p. 1272)), you can modify the **Iterations** entry under **Residual Values**.

In some cases, the maximum residual may occur sometime after the iteration specified in the **Iterations** field. If this should occur, you can click the **Renormalize** button to set the normalization factors for all variables to the maximum values in the residual histories. Subsequent plots and printed reports will use the new normalization factor.

You can also specify the normalization factor (the denominator in *Equation 28–9* (p. 1270) or *Equation 28–13* (p. 1272)) explicitly. To modify the normalization factor for a particular variable, enter a new value in the corresponding **Normalization Factor** field in the *Residual Monitors Dialog Box* (p. 1915).

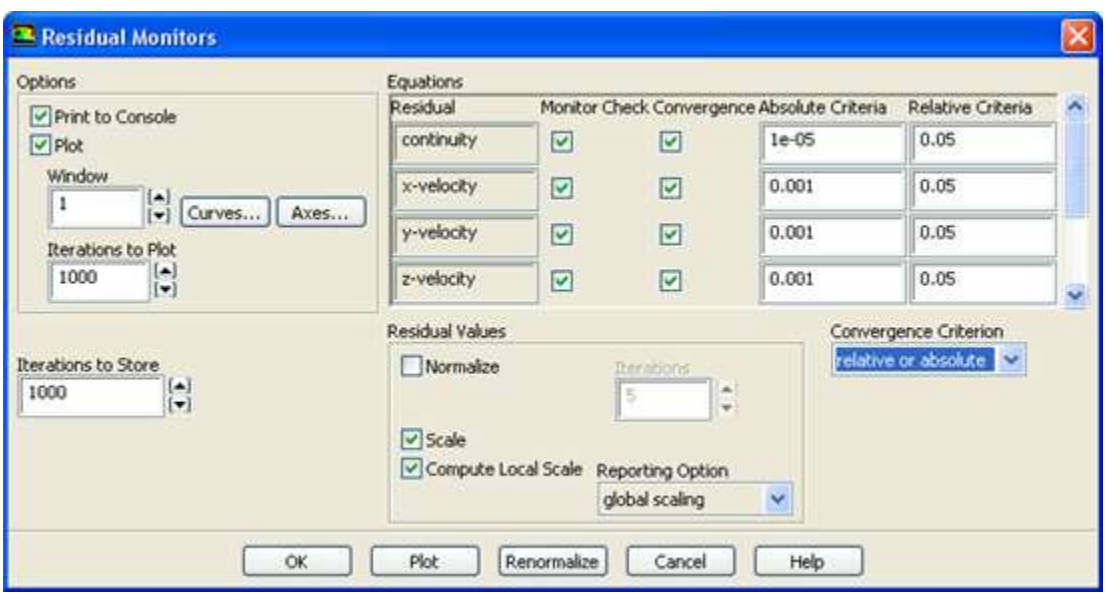
If you wish to report unnormalized, unscaled residuals (*Equation 28–4* (p. 1269) or *Equation 28–10* (p. 1271)), disable the **Normalize** and **Scale** options under **Residual Values** in the *Residual Monitors Dialog Box* (p. 1915). Note that unnormalized, unscaled residuals are stored in the data file regardless of whether the reported residuals are normalized or scaled.

28.15.1.7. Choosing a Convergence Criterion

The ability to choose certain convergence criteria provides you with alternative ways to check convergence when using the iterative transient solver. The various convergence criteria can be selected in the **Residual Monitors** dialog box from the **Convergence Criterion** drop-down list.

Monitors → **Residuals** → **Edit...**

Figure 28.29 The Residual Monitors Dialog Box Displaying Relative or Absolute Convergence



Four options are available for checking an equation for convergence:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

absolute
This is the default. For steady-state cases, **absolute** and **none** are the only options available for selection. The residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a timestep.

relative
The residual of an equation at an iteration of a timestep is compared with the residual at the start of the timestep. If the ratio of the two residuals is less than a user-specified value, that equation is deemed to have converged for a timestep.

relative or absolute
If either the **absolute** convergence criterion or the **relative** convergence criterion is met, the equation is considered converged.

The **Relative Criteria** can be set when **relative** or **relative or absolute** is selected.

none
Convergence checking is disabled.

In many situations, the **absolute** convergence criterion could be too stringent for transient flows causing a large number of iterations per timestep. For example, the scaling of the continuity equation is based on the value of the continuity residual in the first five iterations. The scaling factor could be low if the initial continuity residual is small and thus the scaled residual could fail to meet the **absolute** convergence criterion. With the **relative** convergence criterion, convergence is checked by comparing the residual at an iteration of a timestep with the residual at the beginning of the timestep and hence this problem is alleviated. The **relative or absolute** convergence criterion is useful in situations where the residuals of some of the equations are already very low at the start of a timestep (for example, when a particular variable has reached steady state), and the order of magnitude reduction in residuals is not possible. The **none** option allows you to disable convergence checking by selecting the option in the **Convergence Criterion** drop-down list.

Important

relative and **relative or absolute** convergence criteria are available only with the unsteady pressure-based solver and unsteady density-based solver.

The text command used to access the convergence criterion is

`solve` → `monitors` → `residual` → `criterion-type`

When `criterion-type` is entered, you will have the following choices:

Criterion	Type
0	absolute
1	relative
2	relative or absolute
3	none

For `criterion-type` 1 or 2, the text command `relative-conv-criteria` will appear under the `residual` text menu, where the various `relative-conv-criteria` can be set.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If the NITA solver is enabled, no convergence criteria are available for selection.

28.15.1.8. Modifying Convergence Criteria

Depending on the **Convergence Criterion** you choose, ANSYS FLUENT will check for convergence. If convergence is being monitored, the solution will stop automatically when each variable meets its specified convergence criterion. Convergence checks can be performed only for variables for which you are monitoring residuals (i.e., variables for which the **Monitor** option is enabled).

You can choose whether or not you want to check the convergence for each variable by enabling or disabling the **Check Convergence** option for it in the *Residual Monitors Dialog Box* (p. 1915). To modify the convergence criterion for a particular variable, enter a new value in the corresponding convergence criterion field.

28.15.1.9. Disabling Monitoring

If your problem requires the solution of many equations (e.g., turbulence quantities and multiple species), a plot that includes all residuals may be difficult to read. In such cases, you may choose to monitor only a subset of the residuals, perhaps those that affect convergence the most. You can indicate whether or not you want to monitor residuals for each variable by enabling or disabling the relevant check box in the **Monitor** list of the *Residual Monitors Dialog Box* (p. 1915).

28.15.1.10. Plot Parameters

If you choose to plot the residual values (either interactively during the solution or using the **Plot** button after calculations are complete), there are several display parameters you can modify.

In the **Window** field under **Options**, you can specify the ID of the graphics window in which the plot will be drawn. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to thiswindow to update the residual plot, and then returned to its previous value. Thus, the residual plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

You can modify the number of residual history points to be displayed in the plot by changing the **Iterations to Plot** entry under **Options**. If you specify n points, ANSYS FLUENT will display the last n history points. Since the y axis is scaled by the minimum and maximum values of all points in the plot, you can zoom in on the end of the residual history by setting **Iterations to Plot** to a value smaller than the number of iterations performed. If, for example, the residuals jumped early in the calculation when you turned on turbulence, that peak broadens the overall range in residual values, making the smaller fluctuations later on almost indistinguishable. By setting the value of **Iterations to Plot** so that the plot does not include that early peak, your y-axis range is better suited to the values that you are interested in seeing. For more information on residual history points, please refer to the discussion of storing residual history points, described earlier in this section.

You can also modify the attributes of the plot axes and the residual curves. Click the **Axes...** or **Curves...** button to open the *Axes Dialog Box* (p. 2022) or *Curves Dialog Box* (p. 2024). See *Axes Dialog Box* (p. 2022) and *Curves Dialog Box* (p. 2024) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that entering a value for **Iterations to Plot** does not necessarily mean *solved* iterations but rather *stored* (or sampled) data points. Note also that the frequency of the data storage will diminish towards the start of the solution as the number of solved iterations increases. Due to this, whenever the *stored* iterations is greater than the *solved* iterations, if you plot *n* iterations, you actually see a history that goes back further than *n* solved iterations.

28.15.1.11. Postprocessing Residual Values

If you are having solution convergence difficulties, it is often useful to plot the residual value fields (e.g., using contour plots) to determine where the high residual values are located. When you use one of the density-based solver, the residual values for all solution variables are available in the **Residuals...** category in the postprocessing dialog boxes. (If you read case and data files into ANSYS FLUENT, you will need to perform at least one iteration before the residual values are available for postprocessing.) For the pressure-based solver, however, only the mass imbalance in each cell is available by default.

If you want to plot residual value fields for a pressure-based solver calculation, you will need to do the following:

1. Read in the case and data files of interest (if they are not already in the current session).
2. Use the `expert` command in the `solve/set/` text menu to enable the saving of residual values.

   `solve` → `set` → `expert`

   Among other questions, ANSYS FLUENT will ask if you want to save cell residuals for postprocessing. Enter `yes` or `y`, and keep the default settings for all of the other questions (by pressing the <RETURN> key).
3. Perform at least one iteration.

The solution variables for which residual values are available will appear in the **Residuals...** category in the postprocessing dialog boxes. Note that residual values are *not* available for the radiative transport equations solved by the discrete ordinates radiation model.

28.15.2. Monitoring Statistics

If you are solving a fully-developed periodic flow, you may want to monitor the pressure gradient or the bulk temperature ratio, as discussed in *Periodic Flows* (p. 493).

If you are solving an unsteady flow (especially if you are using the explicit time stepping option), you may want to monitor the "time" that has elapsed during the calculation. The physical time of the flow field starts at zero when you initialize the flow. (See *Performing Time-Dependent Calculations* (p. 1255) for details about modeling unsteady flows.)

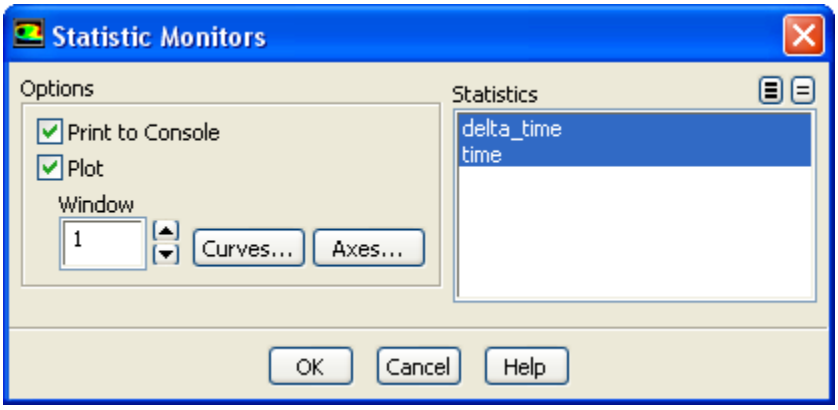
If you are using the adaptive time stepping method described in *Adaptive Time Stepping* (p. 1264), you may want to monitor the size of the time step, Δt.

You can use the *Statistic Monitors Dialog Box* (p. 1917) (*Figure 28.30* (p. 1278)) to print or plot these quantities during the calculation.

Monitors → **Statistic** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.30 The Statistic Monitors Dialog Box



The procedure for setting up this monitor is listed below:

1. Indicate the type of report you want by enabling the **Print to Console** option for a printout and/or the **Plot** option for a plot.
2. Select the appropriate quantity in the **Statistics** list.
3. If you are plotting the quantities, you can set any of the plotting options discussed below.

28.15.2.1. Plot Parameters

If you choose to plot the statistics, there are several display parameters you can modify.

In the **Window** field, you can specify the ID of the graphics window in which the plot will be drawn (or in which the first plot will be drawn, if you are plotting more than one quantity.) When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the plot, and then returned to its previous value. Thus, the statistics plot can be maintained in a separate window that does not interfere with other graphical postprocessing. Note that additional quantities that you have selected in the **Statistics** list will be plotted in windows with incrementally higher IDs.

You can also modify the attributes of the plot axes and curves. Click the **Axes...** or **Curves...** button to open the *Axes Dialog Box* (p. 2022) or *Curves Dialog Box* (p. 2024). See *Axes Dialog Box* (p. 2022) and *Curves Dialog Box* (p. 2024) for details.

28.15.3. Monitoring Force and Moment Coefficients

You can set up your case file so that drag, lift, and moment coefficients are computed and stored at the end of every iteration (for steady-state solutions) or time step (for transient solutions), and thus create a convergence history. You can print and plot this convergence data, and also save it to an external file. The external file is written in the ANSYS FLUENT XY plot file format described in *XY Plot File Format* (p. 1477). Monitoring force coefficients can be useful when you are calculating external aerodynamics, for example, and are especially interested in the lift coefficient. By monitoring these values you may also be able to stop the calculation early and reduce the processing time, as sometimes the force and moment coefficients converge before the residuals have decreased three orders of magnitude. (In such an instance, you should be sure to check the mass flow rate and heat transfer rate as well, to ensure that the mass and energy are being suitably conserved. This is accomplished using the *Flux Reports Dialog Box* (p. 2027), as described in *Fluxes Through Boundaries* (p. 1510).)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The force and moment coefficients are calculated using the reference values entered in the **Reference Values** task page. For information about how these coefficients are calculated, see Computing Forces, Moments, and the Center of Pressure in the Theory Guide.

28.15.3.1. Setting Up Force and Moment Coefficient Monitors

To begin setting up a force or moment monitor, first enter appropriate values in the **Reference Values** task page, as described in *Reference Values* (p. 1523). The relevant values include the following:

- The force coefficients use the reference area, density, and velocity.
- The moment coefficients use the reference area, density, velocity and length.

Next, open the appropriate dialog box using the **Monitors** task page. Select either **Drag**, **Lift**, or **Moment** from the **Residuals, Statistic and Force Monitors** selection list and click the **Edit...** button to open the *Drag Monitor Dialog Box* (p. 1918), *Lift Monitor Dialog Box* (p. 1920), or *Moment Monitor Dialog Box* (p. 1921), respectively (*Figure 28.31* (p. 1280)— *Figure 28.33* (p. 1281)). Note that you can only access one of these monitor dialog boxes at a time, though all three monitors can be used during the same simulation.

Monitors → **Drag** → **Edit...**

Monitors → **Lift** → **Edit...**

Monitors → **Moment** → **Edit...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.31 The Drag Monitor Dialog Box

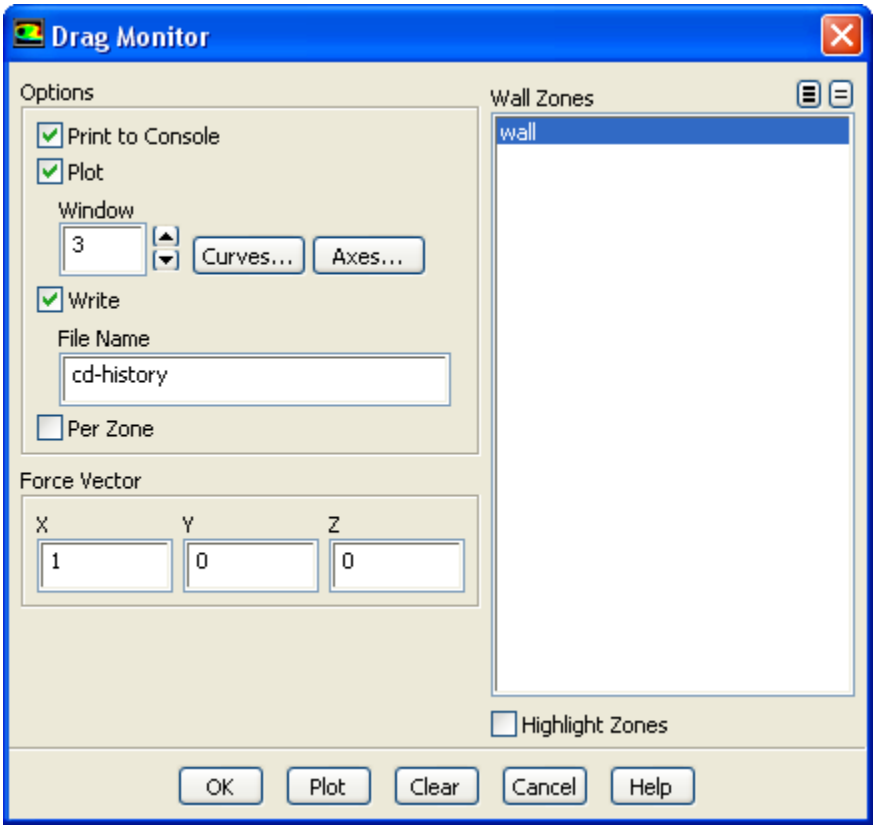
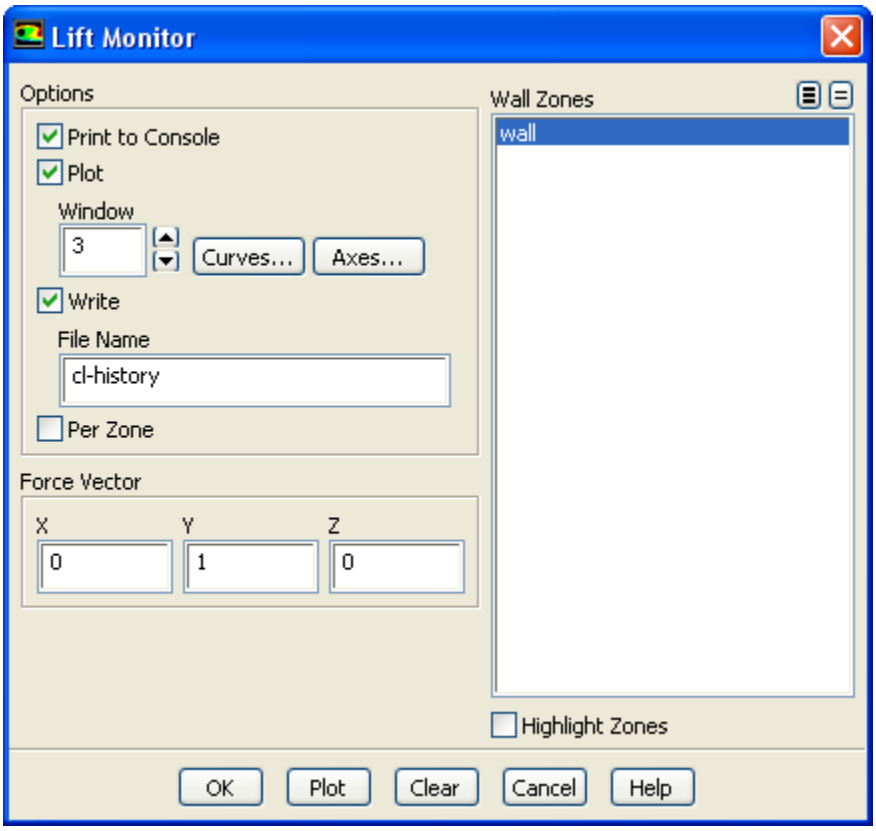
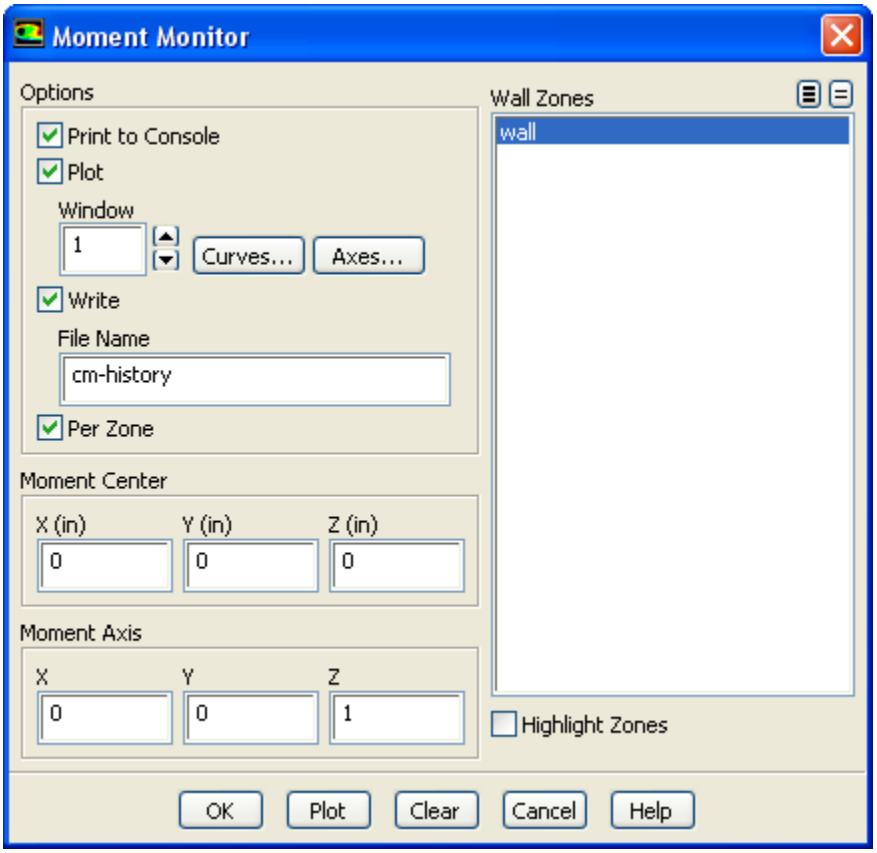


Figure 28.32 The Lift Monitor Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.33 The Moment Monitor Dialog Box



Complete the setup of the force or moment coefficient monitor by performing the following steps in the **Drag Monitor**, **Lift Monitor**, or **Moment Monitor** dialog box:

1. Indicate the method of reporting you want for the data (numerical display, plot, or file), as described in a section that follows.
2. If you want to monitor the force or moment coefficient data from individual wall zones rather than the net results from a group of wall zones, enable the **Per Zone** option. Further details are provided in a section that follows.
3. Depending on the coefficient that will be monitored, perform one of the following steps:
   - In the **Drag Monitor** or **Lift Monitor** dialog box, enter the **X**, **Y**, and **Z** components of the **Force Vector** along which the forces will be computed. By default, the **Force Vector** for the drag coefficient is a unit vector in the x direction, whereas for the lift coefficient it is a unit vector in the y direction.
   - In the **Moment Monitor** dialog box, enter the Cartesian coordinates (**X**, **Y**, and **Z**) of the **Moment Center**, about which moments will be computed. The default **Moment Center** is (**0**, **0**, **0**). You also need to enter the **X**, **Y**, and **Z** components for the **Moment Axis**, along which the moment coefficient will be calculated. By default, the **Moment Axis** is defined as a unit vector in the z direction.
4. Specify the wall zone(s) for which the coefficient(s) will be computed by making selections in the **Wall Zones** selection list.
5. You have the option to highlight a selected boundary zone and have it displayed in the graphics window by enabling **Highlight Zone**.
6. Click **OK** to save the monitor settings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you have set up all of the force and moment coefficient monitors, you can then run the calculation and view the data in the console window, graphics window, or file.

Important

Only the processed force and moment coefficient data is saved. If you decide to change any of the parameters controlling the monitoring (e.g., the reference values, force vector, moment center, moment axis, wall zones) and run further calculations, you may see a discontinuity in the data, because the previous data is not updated to match the new settings. Usually, you will want to delete the previous force and moment coefficient data before continuing to iterate with revised monitor parameters.

28.15.3.1.1. Specifying the Reporting Methods

There are three methods available for reporting the force and moment coefficients. To display the coefficient value(s) in the console window after every iteration or time step, enable the **Print to Console** option in the **Options** group box of the monitor dialog box. To plot the coefficient in the graphics window indicated in the **Window** text box, enable the **Plot** option. If you want to save the data to a file, enable the **Write** option and specify the **File Name**. You can enable any combination of these options simultaneously.

Important

If you choose *not* to save the force or moment coefficient data in a file, this information will be lost when you exit the current ANSYS FLUENT session.

You can display a plot of the coefficient monitor data generated during the last calculation even if the **Plot** option was not enabled during the calculation, as long as either **Print to Console** or **Write** was enabled. Simply click the **Plot** button and the plot will be displayed in the active graphics window (if the **Plot** option *is not* enabled) or in the specified **Window** (if the **Plot** option *is* enabled).

28.15.3.1.1.1. Plot Parameters

If you choose to plot the force or moment coefficients (either by enabling the **Plot** option prior to running the solution or by clicking the **Plot** button after the calculation is complete), there are several display parameters you can modify:

- Under **Window**, you can specify the ID of the graphics window in which the plot for the force or moment coefficient will be displayed.
- You can modify the attributes of the coefficient curves and plot axes for each monitor. Click the **Curves...** or the **Axes...** button to open the *Curves Dialog Box* (p. 2024) or the *Axes Dialog Box* (p. 2022). See *Curves Dialog Box* (p. 2024) and *Axes Dialog Box* (p. 2022) for details.

28.15.3.1.2. Monitoring Individual Walls

By default, ANSYS FLUENT will compute and monitor the sum of the force or moment coefficients for all of the selected walls. If you have selected multiple walls and you want to monitor the force or moment coefficient on each wall separately, you can enable the **Per Zone** option in the **Options** group box in the monitor dialog box. The specified force vector or moment center and axis will apply to all of the selected walls.

If the monitor results are displayed in the console window (using the **Print to Console** option), the force or moment coefficient for each wall zone will be printed in a separate column. If the results are plotted (using the **Plot** option or button), a separate curve for each wall zone will be drawn in the specified graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the results are written to a file (using the **Write** option), the file will be in a tab-separated column format based on the XY plot file format described in *XY Plot File Format* (p. 1477).

28.15.3.1.3. Faster Force and Moment Monitoring

While all the above reporting options (**Print to Console**, **Plot**, and **Write**) are still applicable and can be enabled altogether or in any combination, you can speed up the force and moment monitoring during your simulation. To do this, enable only the **Write** option and make sure that **Print to Console** and **Plot** are disabled. You can enable or disable **Per Zone** as required by your simulation. When only the **Write** option is enabled, fast monitoring wil be used, reducing the overhead in time monitoring. Note that in this case the **Plot** button in the **Drag Monitor**, **Lift Monitor**, or **Moment Monitor** dialog boxes will be inactive. However, you can still plot the force or moment data using the **File XY Plot** dialog box (see *XY Plots of File Data* (p. 1472)).

Plots → **File** → **Set Up...**

28.15.3.1.4. Discarding the Monitor Data

Should you decide that the data gathered by the force or moment monitor is not useful (e.g., if you are restarting the calculation with revised reference values), you can discard the data accumulated during the last calculation by clicking the **Clear** button. All of the data gathered as a result of the settings in that monitor dialog box will be deleted, including the associated file (with the name indicated in the **File Name** text box). When you use the **Clear** button, you will need to confirm the data discard in a *Question Dialog Box* (p. 32). Only the coefficient monitor data is discarded as a result of this operation; the solution data is not affected.

28.15.4. Monitoring Surface Integrals

At the end of each solver iteration or time step, the average, mass average, integral, flow rate, or other integral report of a field variable or function can be monitored on a surface. You can print and plot these convergence data, and also save them in an external file. The external file is written in the ANSYS FLUENT XY plot file format described in *XY Plot File Format* (p. 1477). The report types available are the same as those in the *Surface Integrals Dialog Box* (p. 2031), as described in *Surface Integration* (p. 1519).

Monitoring surface integrals can be used to check for both iteration convergence and mesh independence. For example, you can monitor the average value of a certain variable on a surface. When this value stops changing, you can stop iterating. You can then adapt the mesh and reconverge the solution. The solution can be considered mesh-independent when the average value on the surface stops changing between adaptions.

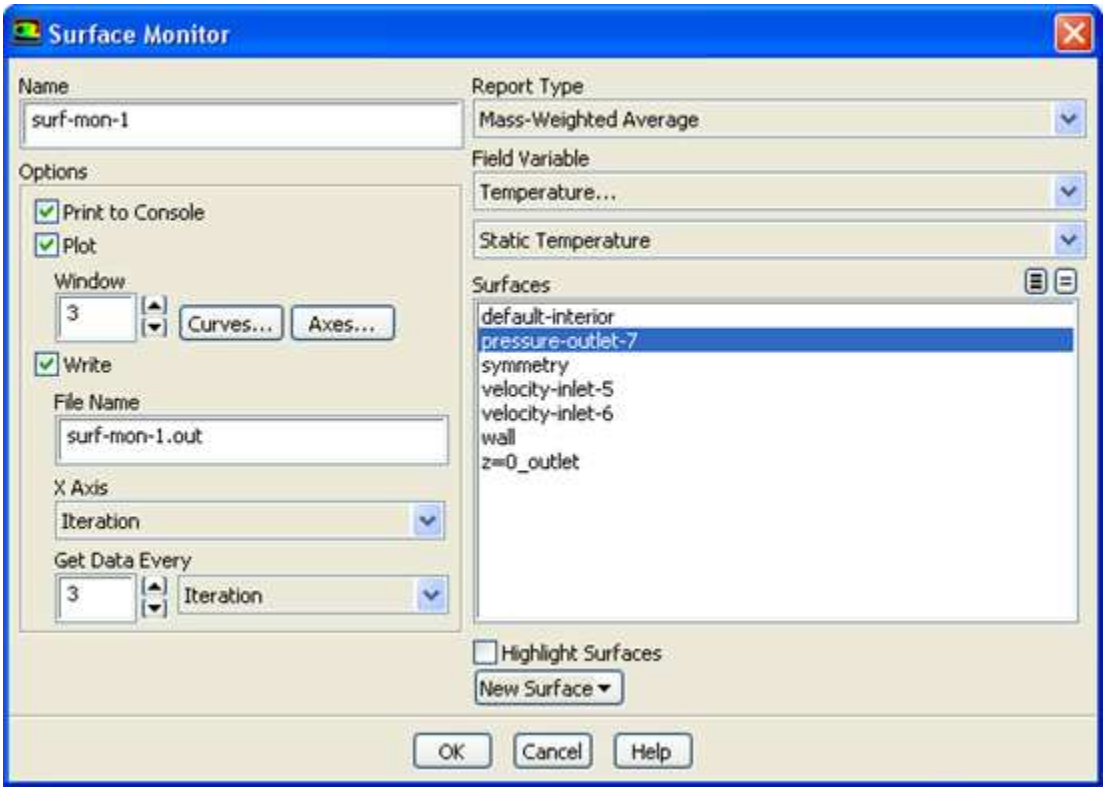
28.15.4.1. Overview of Defining Surface Monitors

You can use the *Surface Monitor Dialog Box* (p. 1923) (*Figure 28.34* (p. 1284)) to create surface monitors and indicate whether and when each one's history is to be printed, plotted, or saved. It also allows you to define what each monitor tracks (i.e., the average, integral, flow rate, mass average, or other integral report of a field variable or function on one or more surfaces).

Monitors (Surface Monitors) → **Create...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.34 The Surface Monitor Dialog Box



The procedure for defining surface monitors is as follows:

1. Enter a name for the monitor under the **Name** heading, and use the **Print to Console**, **Plot**, and **Write** check buttons to indicate the report(s) you want (plot, printout, or save to a file), as described below.
2. If you are plotting the data or writing them to a file, specify the parameter to be used as the x-axis value (the y-axis value corresponds to the monitored data). In the **X Axis** drop-down list, select **Iteration**, **Time Step**, or **Flow Time** as the x-axis function against which monitored data will be plotted or written. **Time Step** and **Flow Time** are valid choices only if you are calculating unsteady flow. If you choose **Time Step**, the x axis of the plot will indicate the time step, and if you choose **Flow Time**, it will indicate the elapsed time.
3. If you are plotting the monitored data, specify the ID of the graphics window in which the plot will be drawn in the **Window** field. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the plot, and then returned to its previous value. Thus, each surface-monitor plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

   In order to have multiple monitors display in a single graphics window, you can set the **Window** ID to correspond to the same ID for different monitors. This is useful when you have multiple monitor displays on the screen, you can set all monitors to the same display. For example, for three different monitors, you can set the **Window** ID to 1 for each of the different monitors in order to display all three monitors in a single window. The name of the monitors (`surf-mon-1.out`, etc.) will be different, but only the **Window** ID will remain the same. So that each monitor has data that is stored in a different file, but the data is displayed in the same window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that surface and volume monitors cannot be displayed in the same window.

Important

If multiple monitors are plotted in the same window, make sure you set an axes range that can be applied to all the monitors. This axes range will be the same for all the monitors in the shared plot window. Otherwise, the most recently defined monitor, sharing the same window as the other monitors, will determine the axes range. If the default option of **Auto Range** (in the **Axes** dialog box) is enabled for all the monitors sharing the same plot window, then the default value for **Minimum** will be the minimum value of all the monitors, and the default **Maximum** will be the maximum value of all the monitors.

Modifying plot attributes can be achieved by clicking the **Curves** and the **Axes** button. See *Modifying Axis Attributes* (p. 1480) *Modifying Curve Attributes* (p. 1482)

4. If you are writing the monitored data to a file, specify the **File Name**.
5. Indicate the frequency at which you want to plot, print, or write the surface monitor by entering a number under **Get Data Every**. A default value of 1 will allow you to monitor at every **Iteration** or **Time Step**. **Time Step** is a valid choice only if you are calculating unsteady flow. If you specify every **Iteration**, and the **Reporting Interval** in the *Run Calculation Task Page* (p. 1955)is greater than 1, the monitor will be updated at every reporting interval instead of at each iteration (e.g., for a reporting interval of 2, the monitor will be updated after every other iteration. If the reporting interval is 2 and monitor frequency is at **Get Data Every** 3**Iteration**s, then the monitoring will be done at multiples of six, which is the least common multiple of the two numbers). If you specify every **Time Step**, the reporting interval will have no effect; the monitor will always be updated after the specified number of time steps.
6. Choose the integration method for the surface monitor by selecting **Integral**, **Standard Deviation**, **Flow Rate**, **Mass Flow Rate**, **Volume Flow Rate**, **Area-Weighted Average**, **Mass-Weighted Average**, **Sum**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Vertex Average**, **Vertex Minimum**, or **Vertex Maximum** from the **Report Type** drop-down list. These methods are described in *Surface Integration* (p. 1519).
7. Specify the variable or function to be integrated in the **Field Variable** drop-down list. First select the desired category in the upper drop-down list. You can then select one of the related quantities in the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
8. In the **Surfaces** list, choose the surface or surfaces on which you wish to integrate.
9. Click **OK** in the **Surface Monitors** dialog box after you finish defining all surface monitors.

Note

If you want to set a monitor for an iso-surface, and the iso-surface is dependent on solver data e.g. velocity, pressure, custom field function etc., make sure to set the monitor after initializing or reading the data file.

28.15.4.2. Printing, Plotting, and Saving Surface Integration Histories

There are three methods available for reporting the selected surface integration. To print the surface integration in the console window after each iteration, enable the **Print to Console** option in the **Surface Mon-**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

itor dialog box. To plot the integrated values in the graphics window, enable the **Plot** option. If you want to save the values to a file, enable the **Write** option and specify the **File Name**. You can enable any combination of these options simultaneously.

Important

If you choose *not* to save the surface integration data to a file, this information will be lost when you exit the current ANSYS FLUENT session.

28.15.4.2.1. Plot Parameters

You can modify the attributes of the plot axes and curves used for each surface-monitor plot. Click the **Axes...** or **Curves...** button in the *Surface Monitor Dialog Box* (p. 1923) to open the *Axes Dialog Box* (p. 2022) or *Curves Dialog Box* (p. 2024) for that surface-monitor plot. See *Axes Dialog Box* (p. 2022) and *Curves Dialog Box* (p. 2024) for details.

28.15.5. Monitoring Volume Integrals

At the end of each solver iteration or time step, the volume or the sum, volume integral, volume average, mass integral, or mass average of a field variable or function can be monitored in one or more cell zones. You can print and plot these convergence data, and also save them in an external file. The external file is written in the ANSYS FLUENT XY plot file format described in *XY Plot File Format* (p. 1477). The report types available are the same as those in the *Volume Integrals Dialog Box* (p. 2034), as described in *Volume Integration* (p. 1521).

Monitoring volume integrals can be used to check for both iteration convergence and mesh independence. For example, you can monitor the average value of a certain variable in a particular cell zone. When this value stops changing, you can stop iterating. You can then adapt the mesh and reconverge the solution. The solution can be considered mesh-independent when the average value in the cell zone stops changing between adaptions.

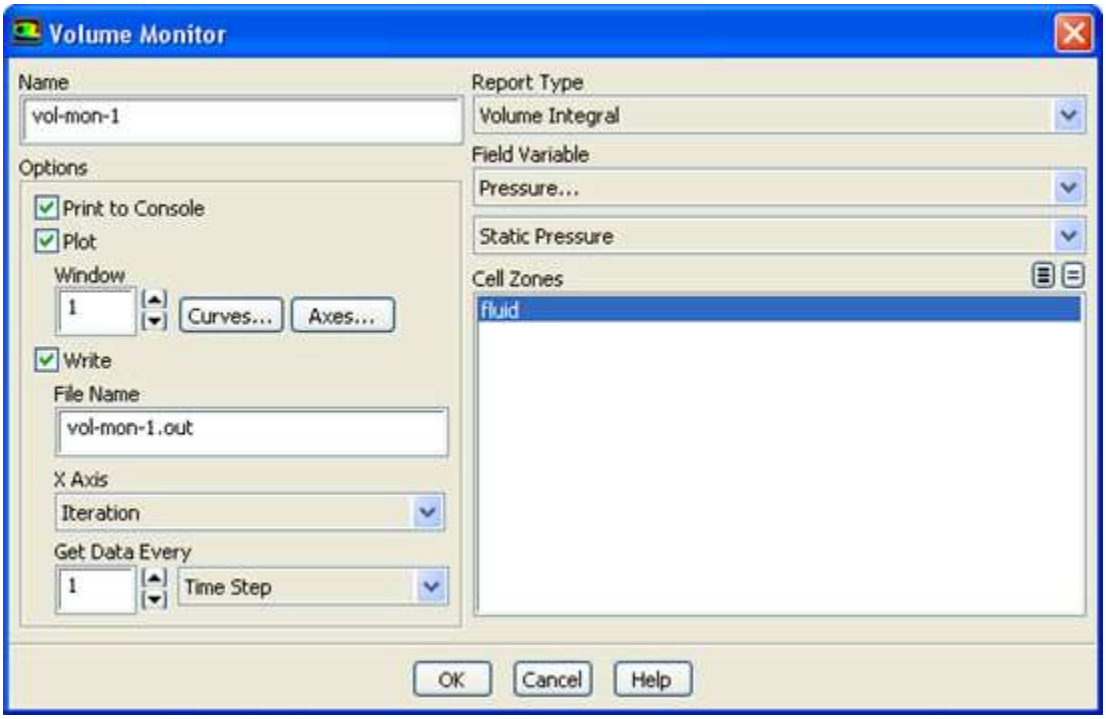
28.15.5.1. Overview of Defining Volume Monitors

You can use the *Volume Integrals Dialog Box* (p. 2034) (*Figure 28.35* (p. 1287)) to create volume monitors and indicate whether and when each one's history is to be printed, plotted, or saved. You can also define what each monitor tracks (i.e., the volume or the sum, integral, or average of a field variable or function in one or more cell zones).

Monitors (Volume Monitors) → **Create...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.35 The Volume Monitor Dialog Box



The procedure for defining volume monitors is as follows:

1. Enter a name for the monitor under the **Name** heading, and use the **Plot**, **Print to Console**, and **Write** check buttons to indicate the report(s) you want (plot, print out, or file), as described below.
2. If you are plotting the data or writing them to a file, specify the parameter to be used as the x-axis value (the y-axis value corresponds to the monitored data). In the **X Axis** drop-down list, select **Iteration**, **Time Step**, or **Flow Time** as the x-axis function against which monitored data will be plotted or written. **Time Step** and **Flow Time** are valid choices only if you are calculating unsteady flow. If you choose **Time Step**, the x axis of the plot will indicate the time step, and if you choose **Flow Time**, it will indicate the elapsed time.
3. If you are plotting the monitored data, specify the ID of the graphics window in which the plot will be drawn in the **Window** field. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the plot, and then returned to its previous value. Thus, each volume-monitor plot can be maintained in a separate window that does not interfere with other graphical postprocessing.
4. If you are writing the monitored data to a file, specify the **File Name**.
5. Indicate the frequency at which you want to plot, print, or write the volume monitor by entering a number under **Get Data Every**. A default value of 1 will allow you to monitor at every **Iteration** or **Time Step**.
6. Indicate whether you want to update the monitor every **Iteration** or every **Time Step** by selecting the appropriate item in the drop-down list. **Time Step**is a valid choice only if you are calculating unsteady flow. If you specify every **Iteration**, and the **Reporting Interval** in the *Run Calculation Task Page* (p. 1955)is greater than 1, the monitor will be updated at every reporting interval instead of at each iteration (e.g., for a reporting interval of 2, the monitor will be updated after every other iteration. If the reporting interval is 2 and monitor frequency is at **Get Data Every** 3**Iteration**s, then the monitoring will be done at multiples of six, which is the least common multiple of the two numbers). If you specify

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

every **Time Step**, the reporting interval will have no effect; the monitor will always be updated after the specified number of time steps.

7. Choose the integration method for the volume monitor by selecting **Volume**, **Sum**, **Max**, **Min**, **Volume Integral**, **Volume-Average**, **Mass Integral**, or **Mass-Average** in the **Report Type** drop-down list. These methods are described in *Volume Integration* (p. 1521).
8. Specify the variable or function to be integrated in the **Field Variable** drop-down list. First select the desired category in the upper drop-down list. You can then select one of the related quantities in the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
9. In the **Cell Zones** list, choose the cell zone(s) on which you wish to integrate.
10. Remember to click **OK** in the **Volume Monitors** dialog box after you finish defining all volume monitors.

28.15.5.2. Printing, Plotting, and Saving Volume Integration Histories

There are three methods available for reporting the selected volume integration. To print the volume integration in the console window after each iteration, Enable the **Print to Console** option in the **Volume Monitor** dialog box. To plot the integrated values in the graphics window indicated in **Window**, enable the **Plot** option. If you want to save the values to a file, enable the **Write** option and specify the **File Name**. You can enable any combination of these options simultaneously.

Important

If you choose *not* to save the volume integration data to a file, this information will be lost when you exit the current ANSYS FLUENT session.

28.15.5.2.1. Plot Parameters

You can modify the attributes of the plot axes and curves used for each volume-monitor plot. Click the **Axes...** or **Curves...** button in the *Volume Integrals Dialog Box* (p. 2034) for the appropriate monitor to open the *Axes Dialog Box* (p. 2022) or *Curves Dialog Box* (p. 2024) for that volume-monitor plot. See *Axes Dialog Box* (p. 2022) and *Curves Dialog Box* (p. 2024) for details.

28.16. Executing Commands During the Calculation

As described in *Monitoring Solution Convergence* (p. 1268) and *Animating the Solution* (p. 1296), respectively, you can report and monitor various quantities (e.g., residuals, force coefficients) and create animations of the solution while the solver is performing calculations. ANSYS FLUENT also includes a feature that allows you to define your own command(s) to be executed during the calculation at specified intervals. For example, you can ask ANSYS FLUENT to perform gradient adaption after a set number of iterations. You will specify a series of text commands or use the GUI to define the steps to be performed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that the **Calculation Activities** task page provides options to perform the following during the calculation:

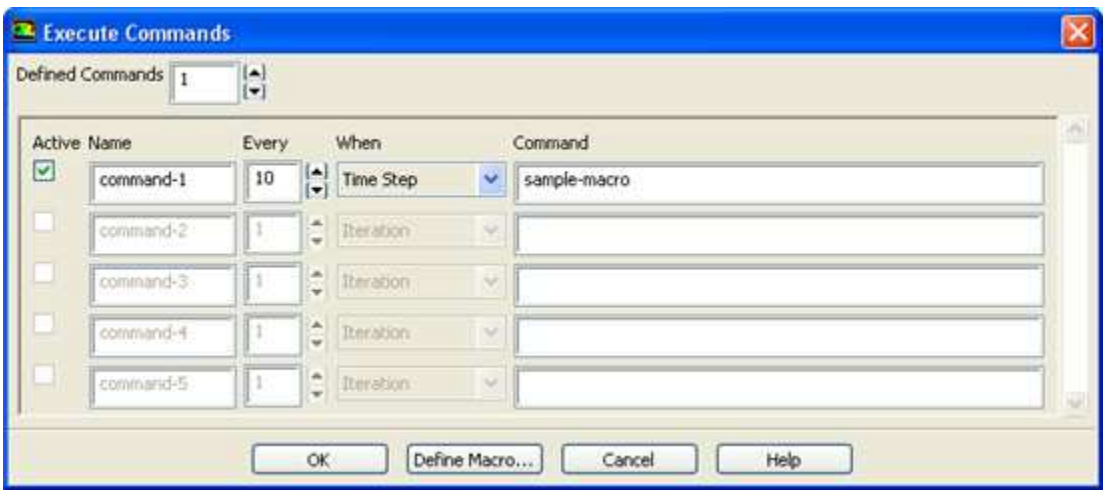
- save case and data files
- export transient solution files
- export transient particle history data files

Each of these options has their own dialog box, which should be used rather than executing a command to perform them. See *Automatic Saving of Case and Data Files* (p. 65) and *Exporting Data During a Transient Calculation* (p. 98) for details.

You will indicate the command(s) that you want the solver to execute at specified intervals during the calculation using the *Execute Commands Dialog Box* (p. 1950) (*Figure 28.36* (p. 1289)).

Calculation Activities (Execute Commands) → Create/Edit...

Figure 28.36 The Execute Commands Dialog Box



The procedure is as follows:

1. Increase the **Defined Commands** value to the number of commands you wish to specify. As this value is increased, additional command entries will become editable. For each command, you will perform the following steps.
2. Enable the **Active** check button next to the command if you want it to be executed during the calculation. You may define multiple commands and choose to use only a subset of them by turning off the check button for those that you do not wish to use.
3. Enter a name for the command under the **Name** heading.
4. Indicate how often you want the command to be executed by setting the interval under **Every** and selecting **Iteration** or **Time Step** in the drop-down list below **When**. (**Time Step** is a valid choice only if you are calculating unsteady flow.) For example, to execute the command every 10 iterations, you would enter `10` under **Every** and select **Iteration** under **When**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you specify an interval in iterations, be sure to keep the **Reporting Interval** in the *Run Calculation Task Page* (p. 1955)at its default value of 1.

5. Define the command by entering a series of text commands in the **Command** field, or by entering the name of a command macro you have defined (or will define) as described in *Defining Macros* (p. 1290).

Important

If the command to be executed involves saving a file, see *Saving Files During the Calculation* (p. 1291) for important information.

For additional information, please see the following sections:

28.16.1. Defining Macros
28.16.2. Saving Files During the Calculation

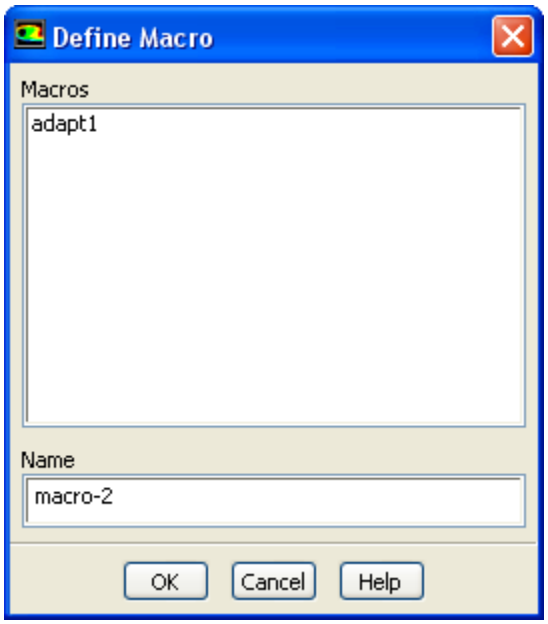
28.16.1. Defining Macros

Macros that you define for automatic execution during the calculation can also be used interactively by you during the problem setup or postprocessing. For example, if you define a macro that performs a certain type of adaption after each iteration, you can also use the macro to perform this adaption interactively.

Definition of a macro is accomplished as follows:

1. In the *Execute Commands Dialog Box* (p. 1950), click the **Define Macro...** button to open the *Define Macro Dialog Box* (p. 1951) (*Figure 28.37* (p. 1290)). Since this is a "modal" dialog box, the solver will not allow you to do anything else until you perform step 2, below.

Figure 28.37 The Define Macro Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Define Macro** dialog box, specify a **Name** for the macro (e.g., `adapt1`) and click **OK**. (The **Define Macro...** button in the **Execute Commands** dialog box will become the **End Macro** button.)
3. Perform the steps that you want the macro to perform. For example, if you want the macro to perform gradient adaption, open the **Gradient Adaption** dialog box, specify the appropriate adaption function and parameters, and click **Adapt** to perform the adaption.

   Important

   If the command to be executed involves saving a file, see *Saving Files During the Calculation* (p. 1291) for important information.

4. When you have completed the steps you wish the macro to perform, click the **End Macro** button in the **Execute Commands** dialog box.

As noted above, once you have defined a macro for execution during the calculation, you can use it at any time. If you defined the macro called `adapt1` to adapt based on pressure gradient, you can simply type `adapt1` in the console window to perform this adaption. This macro is independent of any text menus, so you need not move to a different text menu to use it. Macros can be saved to and read from files. To save all macros that are currently defined, use the `file/write-macros` text command. To read all the macros in a macro file, use the `file/read-macros` text command.

Important

A macro, like a journal file, is a simple record/playback function. It will therefore know nothing about the state in which it was recorded or the state in which it is being played back. You must be careful not to change directories while defining a macro. Also, you must be careful that all surfaces, variables, etc. that are used by the macro have been properly defined when you (or ANSYS FLUENT) invoke the macro.

28.16.2. Saving Files During the Calculation

If the command to be executed during the calculation involves saving a file, you should include a special character in the file name when you enter it in *The **Select File** Dialog Box* (p. 33) so that the solver will know to assign a new name to each file it saves. See *Automatic Numbering of Files* (p. 61) for details about these special characters for filenames.

Important

Note that the **Calculation Activities** task page provides options to perform the following during the calculation:

- save case and data files
- export transient solution files
- export transient particle history data files

Each of these options has their own dialog box, which should be used rather than the **Execute Commands** dialog box. See *Automatic Saving of Case and Data Files* (p. 65) and *Exporting Data During a Transient Calculation* (p. 98) for details.

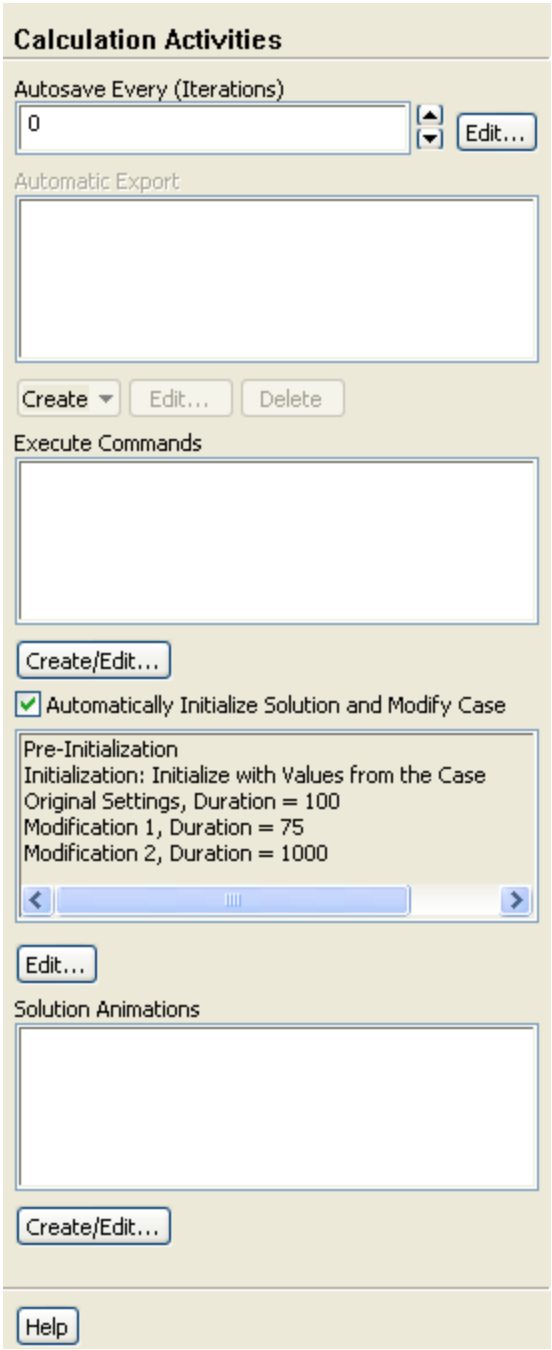
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.17. Automatic Initialization of the Solution and Case Modification

While running a case manually, you can perform certain activities which may facilitate convergence. These actions may take place before initialization, after initialization and/or at other points during the calculation. The process described in this section allows you to enter text commands at each of these times when a case is run from the **Run Calculation** task page, Workbench, or in batch. In the **Calculation Activities** task page, enable **Automatically Initialize Solution and Modify Case** to automatically modify the case.

Calculation Activities → **Automatically Initialize Solution and Modify Case** → **Edit...**

Figure 28.38 The Automatically Initialize Solution and Modify Case Option



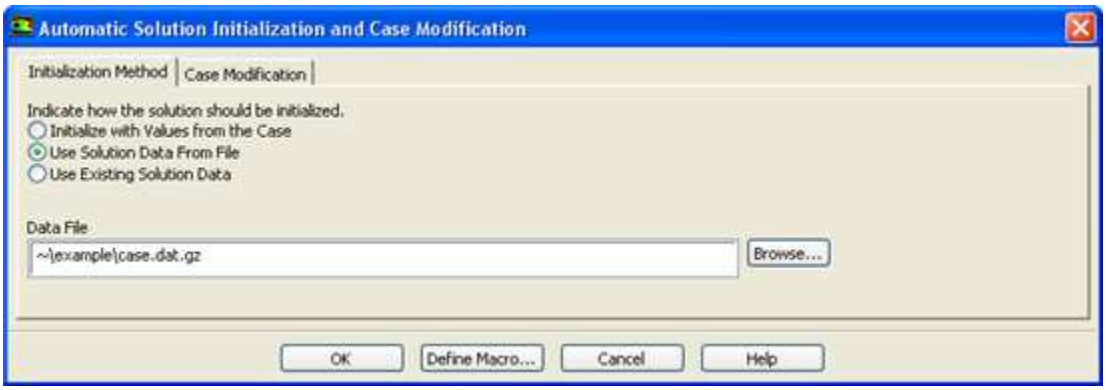
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using this option, you can edit the calculation settings. Note that the original settings always exist and cannot be deleted. The duration of the calculation is defined, so immediately after enabling **Automatically Initialize Solution and Modify Case**, you will notice that the **Calculation Activities** task page reports that the case will be run with the original settings for a single iteration.

You can now control the number of iterations or time steps for the calculation. When **Automatically Initialize Solution and Modify Case** option is disabled, you will have to specify the iterations or time steps using the **Run Calculation** task page.

For an uninitialized case, clicking the **Edit...** button will display the **Automatic Solution Initialization and Case Modification** dialog box, which allows you to specify the initialization method and to modify the case.

Figure 28.39 The Automatic Solution Initialization and Case Modification Dialog Box



In the **Initialization Method** tab, you can specify four different initialization methods:

Initialize with Values from the Case
uses the values set in the **Solution Initialization** task page.

Use Solution Data from File
requires you to read in a data file containing the desired initialization for this case, as shown in *Figure 28.39* (p. 1293).

Use Existing Solution Data
is analogous to changing the values in a case and continuing the calculation. However, the iteration counter will be reset to 0 so that the modifications can be applied. Use this method when no solution data exists, similar to the first run.

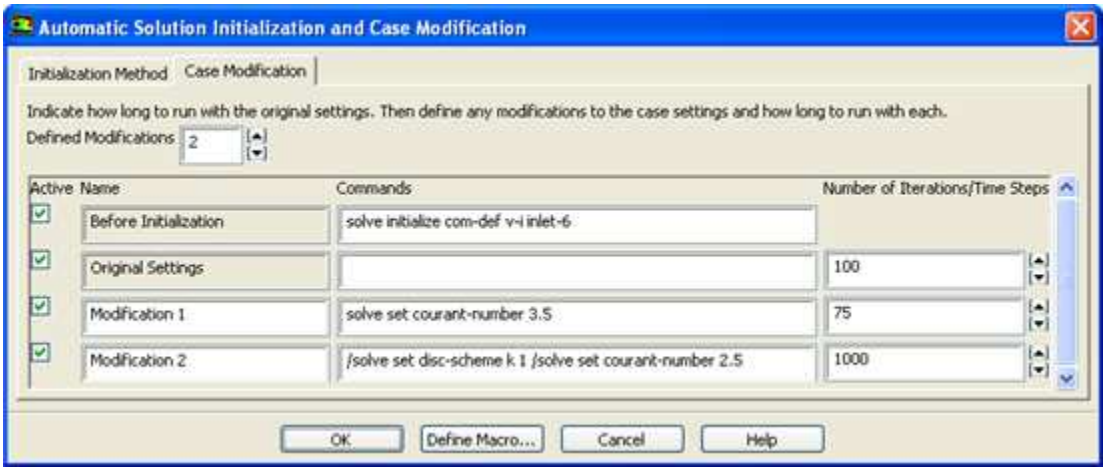
Important

Whenever the case is initialized, the iteration count is set to 0.

In the **Case Modification** tab, you can indicate how long you would like to run with the original settings, then make any modifications to the case settings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.40 The Case Modification Tab



If you decide to make no modifications, then the counter in the **Defined Modifications** box will be set to 0. However, you still have the option to specify settings that you would like to apply before initialization, or you can change your original settings. If **Before Initialization** is enabled, then you can type the text commands in the **Commands** field. If **Original Settings** is enabled, you can type text commands in the **Commands** field, and/or specify the **Number of Iterations/Time Steps**. When entering more than one text command in a single **Commands** field, begin each text command with a `/`, as shown in *Figure 28.40* (p. 1294).

Important

Note that some text commands require arguments (e.g., numbers, file names, yes/no responses), which are requested through follow-up prompts in the console. When entering such text commands in the **Commands** field of the **Case Modification** tab, be sure to include your responses to the prompts. If you do not include these responses, the associated command will not be executed (unless it is a yes/no question, in which case ANSYS FLUENT will use `yes` by default).

If you decide to run the calculation further with modifications to your case, increase the number of **Defined Modifications** and specify additional commands. The settings shown in *Figure 28.40* (p. 1294) result in the following actions:

1. The default initial values for initialization are computed from the velocity-inlet zone **inlet-6**.
2. The iteration count is set to 0 (in all situations) and the solution is initialized.
3. The calculation is run for 100 iterations or until convergence.
4. The Courant number is set to 3.5.
5. The calculation continues for 75 iterations or until convergence.
6. The discretization scheme for turbulent kinetic energy is set to second order upwind, and the Courant number is set to 2.5.
7. The calculation continues for 1000 iterations or until convergence.

You can make the above changes sequentially and run your case, or you can specify them all at once. When you have completed making the modifications, click **OK**. A **Question** dialog box may appear, prompting you to take specific actions. For example, if the **Original Settings** field is empty, then you may be notified that the original settings will be lost if the case is saved after the modifications are applied. It will prompt you for a response when asked if you would like to add commands that specify the original settings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

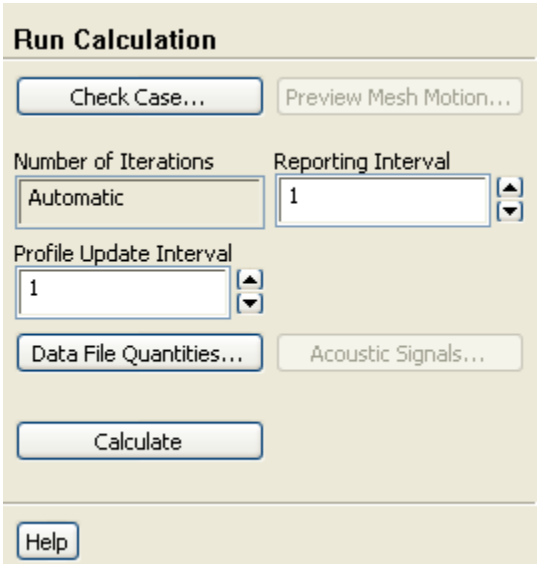
If you specify commands for the **Original Settings**, they will be applied to the case before the first iteration/time step.

Your actions will be summarized in the **Calculation Activities** task page, as shown in *Figure 28.38* (p. 1292).

If you disable **Automatically Initialize Solution and Modify Case**, the settings will be disabled and retained, but will not be applied to the case.

When **Automatically Initialize the Solution and Modify the Case** is enabled, settings defined in the **Run Calculation** task page will be ignored. Instead, the **Number of Iterations** will be defined as **Automatic**, as shown in *Figure 28.41* (p. 1295).

Figure 28.41 The Run Calculation Task Page



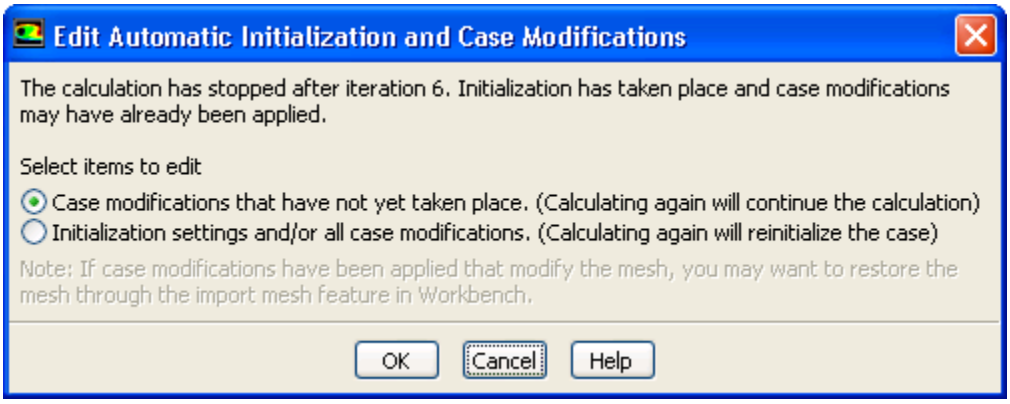
For additional information, please see the following sections:
28.17.1. Altering the Solution Initialization and Case Modification after Calculating

28.17.1. Altering the Solution Initialization and Case Modification after Calculating

If you decide to edit the solution initialization and case modification settings and one or more iterations have been calculated, then clicking the **Edit...** button for the **Automatically Initialize Solution and Modify Case** option will open the **Edit Automatic Initialization and Case Modifications** dialog box, as shown in *Figure 28.42* (p. 1296).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.42 The Edit Automatic Initialization and Case Modifications Dialog Box



If you select the first option, the initialization controls and modifications that have already taken place are disabled, therefore you can edit the case modifications that have yet to take place.

If you select the second option, all controls in the **Automatic Solution Initialization and Case Modification** dialog box are enabled and therefore, you can modify any of the settings.

28.18. Animating the Solution

During the calculation, you can have ANSYS FLUENT create an animation of contours, vectors, XY plots, monitor plots (residual, statistic, force, surface, or volume), or the mesh (useful primarily for moving mesh simulations). Before you begin the calculation, you will specify and display the variables and types of plots you want to animate, and how often you want plots to be saved. At the specified intervals, ANSYS FLUENT will display the requested plots, and store each one. When the calculation is complete, you can play back the animation sequence, modify the view (for mesh, contour, and vector plots), if desired, and save the animation to a series of picture files or an MPEG file

Instructions for defining a solution animation sequence are provided in *Defining an Animation Sequence* (p. 1296). *Playing an Animation Sequence* (p. 1300) describes how to play back and save the animation sequences you have created, and how to read a previously-saved animation sequence into ANSYS FLUENT.

For additional information, please see the following sections:

28.18.1. Defining an Animation Sequence
28.18.2. Playing an Animation Sequence
28.18.3. Saving an Animation Sequence
28.18.4. Reading an Animation Sequence

28.18.1. Defining an Animation Sequence

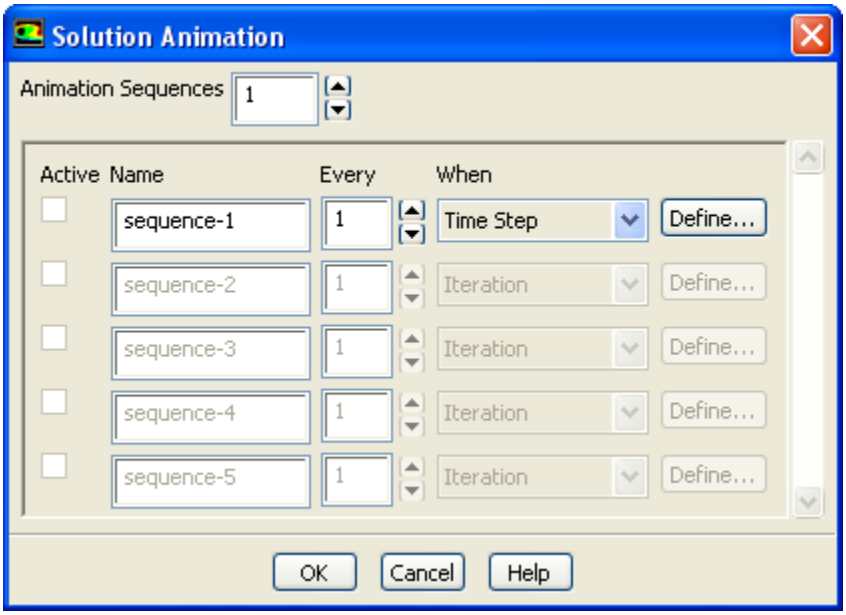
You can use the *Solution Animation Dialog Box* (p. 1953) (*Figure 28.43* (p. 1297)) to create an animation sequence and indicate how often each frame of the sequence should be created. The *Animation Sequence Dialog Box* (p. 1953) (*Figure 28.44* (p. 1298)), opened from the **Solution Animation** dialog box, allows you to define what each sequence displays (e.g., contours or vectors of a particular variable), where it is displayed, and how each frame is stored.

You will begin the animation sequence definition in the *Solution Animation Dialog Box* (p. 1953) (*Figure 28.43* (p. 1297)).

Calculation Activities (Solution Animations) → Create/Edit...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.43 The Solution Animation Dialog Box

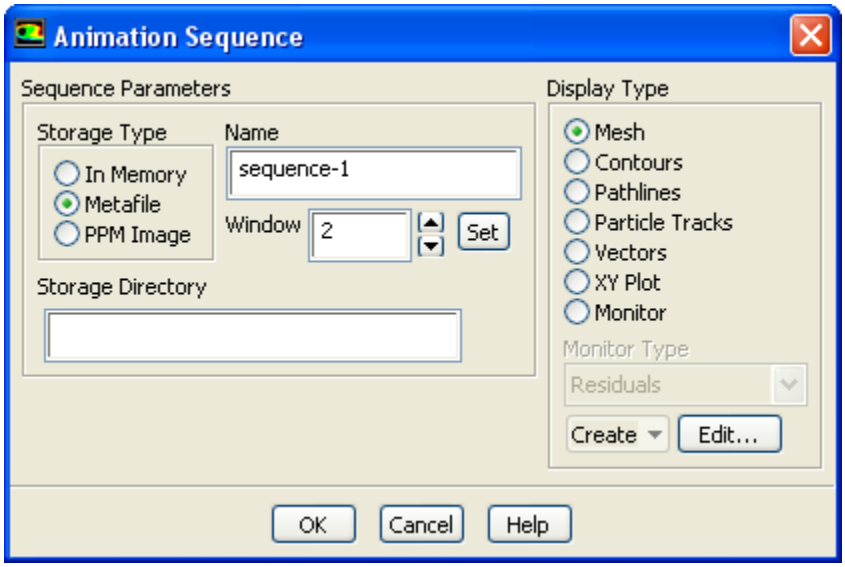


The procedure is as follows:

1. Increase the **Animation Sequences** value to the number of animation sequences you wish to specify. As this value is increased, additional sequence entries in the dialog box will become editable. For each sequence, you will perform the following steps.
2. Enter a name for the sequence under the **Name** heading. This name will be used to identify the sequence in the **Playback** dialog box, where you can play back the animation sequences that you have defined or read in. This name will also be used as the prefix for the file names if you save the sequence frames to disk.
3. Indicate how often you want to create a new frame in the sequence by setting the interval under **Every** and selecting **Iteration** or **Time Step** in the drop-down list below **When**. (**Time Step**is a valid choice only If you are calculating unsteady flow.) For example, to create a frame every10 time steps, you would enter `10` under **Every** and select **Time Step** under **When**.
4. Click the **Define...** button to open the *Animation Sequence Dialog Box* (p. 1953) (*Figure 28.44* (p. 1298)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.44 The Animation Sequence Dialog Box



5. Define the **Sequence Parameters** in the **Animation Sequence** dialog box.

   a. Specify whether you want ANSYS FLUENT to save the animation sequence frames in memory or on your computer's hard drive. To save the animation sequence in memory, select **In Memory** under **Storage Type**. To save the animation sequence to your computer's hard drive as a graphics metafile, select **Metafile** under **Storage Type**. To save the animation sequence to your computer's hard drive as a pixmap image, select **PPM Image** under **Storage Type**.

   Important

   Note that the ANSYS FLUENT metafiles created for each frame in the animation sequence contain information about the entire scene, not just the view that is displayed in the plot. As a result, they can be quite large. By default, the files will be stored to disk. If you do not want to use up disk space to store them, you can instead choose to store them in memory. Storing them in memory will, however, reduce the amount of memory available to the solver. Note that the playback of a sequence stored in memory will be faster than one stored to disk.

   Important

   An advantage to saving the animation sequence using the **PPM Image** option is that you can use the separate pixmap image files for the creation of a single GIF file. GIF file creation can be done quickly with graphics tools provided by other third-party graphics packages such as ImageMagick, i.e., `animate` or `convert`. For example, if you save the PPM files starting with the string `sequence-2`, and you are using the ImageMagick software, you can use the `convert` command with the `-adjoin` option to create a single GIF file out of the sequence using the following command.

```
convert -adjoin sequence-2_00*.ppm sequence2.gif
```

   b. If you selected **Metafile** or **PPM Image** under **Storage Type**, specify the directory where you want to store the files in the **Storage Directory** field. (This can be a relative or absolute path.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Specify the ID of the graphics window where you want the plot to be displayed in the **Window** field, and click **Set**. (The specified window will open, if it is not already open.)

When ANSYS FLUENT is iterating, the active graphics window is set to this window to update the plot. If you want to maintain each animation in a separate window, specify a different **Window** ID for each.

6. Define the display properties for the sequence.

a. Under **Display Type** in the **Animation Sequence** dialog box, choose the type of display you want to animate by selecting **Mesh**, **Contours**, **Pathlines**, **Particle Tracks**, **Vectors**, **XY Plot**, or **Monitor**. If you choose **Monitor**, you can select any of the available types of monitor plots in the **Monitor Type** drop-down list: **Residuals**, **Drag**, **Lift**, **Moment**, or **Statistics**. Furthermore, you can create a **Surface Monitor**, or **Volume Monitor** by selecting it from the **Create**drop-down button.

The first time that you select **Contours**, **Vectors**, or **XY Plot**, or one of the monitor types if you select **Monitor**, ANSYS FLUENT will open the corresponding dialog box (e.g., the *Contours Dialog Box* (p. 1967) or the *Vectors Dialog Box* (p. 1970)) so you can modify the settings and generate the display. To make subsequent modifications to the display settings for any of the display types, click the **Edit...** button to open the dialog box for the selected **Display Type**.

b. Define the display in the dialog box for the selected **Display Type** (e.g., the **Contours** or **Solution XY Plot** dialog box), and click **Display** or **Plot**.

Important

You must click **Display** or **Plot** to initialize the scene to be repeated during the calculation.

See below for guidelines on defining display properties for mesh, contour, and vector displays.

7. Remember to click **OK** in the **Solution Animation** dialog box after you finish defining all animation sequences.

Note that, when you click **OK** in the **Animation Sequence** dialog box for a sequence, the **Active** button for that sequence in the **Solution Animation** dialog box will be turned on automatically. You can choose to use a subset of the sequences you have defined by turning off the **Active** button for those that you currently do not wish to use.

28.18.1.1. Guidelines for Defining an Animation Sequence

If you are defining an animation sequence containing mesh, contour, or vector displays, note the following when you are defining the display:

- If you want to include lighting effects in the animation frames, be sure to define the lights before you begin the calculation. See *Adding Lights* (p. 1423) for information about adding lights to the display.
- If you want to maintain a constant range of colors in a contour or vector display, you can specify a range explicitly by turning off the **Auto Range** option in the **Contours** or **Vectors** dialog box. See *Specifying the Range of Magnitudes Displayed* (p. 1390) or *Specifying the Range of Magnitudes Displayed* (p. 1397 for details.
- Scene manipulations that are specified using the **Scene Description** dialog box will *not* be included in the animation sequence frames. View modifications such as mirroring across a symmetry plan *will* be included.

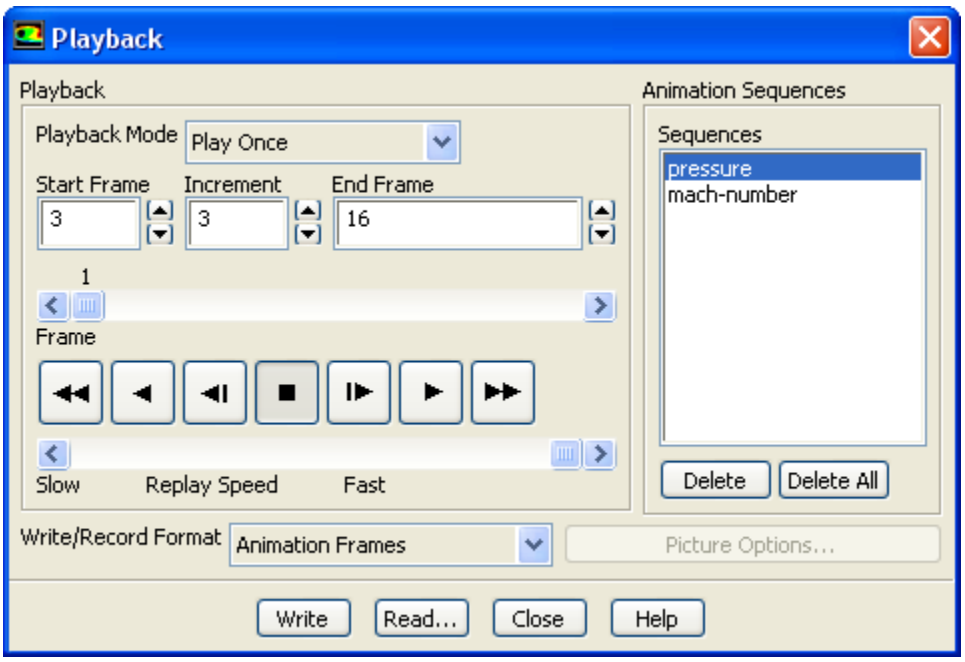
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.18.2. Playing an Animation Sequence

Once you have defined a sequence (as described in *Defining an Animation Sequence* (p. 1296)) and performed a calculation, or read in a previously created animation sequence (as described in *Reading an Animation Sequence* (p. 1303)), you can play back the sequence using the *Playback Dialog Box* (p. 1990) (*Figure 28.45* (p. 1300)).

Graphics and Animation → **Solution Animation Playback** → **Set Up...**

Figure 28.45 The Playback Dialog Box



Under **Animation Sequences** in the **Playback** dialog box, select the sequence you want to play in the **Sequences** list. To play the animation once through from start to finish, click the "play" button under the **Playback** heading. (The buttons function in a way similar to those on a standard video cassette player. "Play" is the second button from the right—a single triangle pointing to the right.) To play the animation backwards once, click the "play reverse" button (the second from the left—a single triangle point to the left). As the animation plays, the **Frame** scale shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If, instead of playing the complete animation sequence, you want to jump to a particular frame, move the **Frame** slider bar to the desired frame number, and the frame corresponding to the new frame number will be displayed in the graphics window.

Important

For smoother animations, enable the **Double Buffering** option in the *Display Options Dialog Box* (p. 1992) (see *Modifying the Rendering Options* (p. 1425)). This will reduce screen flicker during graphics updates.

Additional options for playing back animations are described below.

28.18.2.1. Modifying the View

If you want to replay the animation sequence with a different view of the scene, you can use your mouse to modify (e.g., translate, rotate, zoom) it in the graphics window where the animation is displayed. Note

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that any changes you make to the view for an animation sequence will be lost when you select a new sequence (or reselect the current sequence) in the **Sequences** list.

28.18.2.2. Modifying the Playback Speed

Different computers will play the animation sequence at different speeds, depending on the complexity of the scene and the type of hardware used for graphics. You may want to slow down the playback speed for optimal viewing. Move the **Replay Speed** slider bar to the left to reduce the playback speed (and to the right to increase it).

28.18.2.3. Playing Back an Excerpt

You may sometimes want to play only one portion of a long animation sequence. To do this, you can modify the **Start Frame** and the **End Frame** under the **Playback** heading. For example, if your animation contains 50 frames, but you want to play only frames 20 to 35, you can set **Start Frame** to 20 and **End Frame** to 35. When you play the animation, it will start at frame 20 and finish at frame 35.

28.18.2.4. "Fast-Forwarding" the Animation

You can "fast-forward" or "fast-reverse" the animation by skipping some of the frames during playback. To fast-forward the animation, you will need to set the **Increment** and click the fast-forward button (the last button on the right—two triangles pointing to the right). If, for example, your **Start Frame** is 1, your **End Frame** is 15, and your **Increment** is 2, when you click the fast-forward button, the animation will show frames 1, 3, 5, 7, 9, 11, 13 and 15. Clicking on the fast-reverse button (the first button on the left—two triangles pointing to the left) will show frames 15, 13, 11,...1.

28.18.2.5. Continuous Animation

If you want the playback of the animation to repeat continuously, there are two options available. To continuously play the animation from beginning to end (or from end to beginning, if you use one of the reverse play buttons), select **Auto Repeat** in the **Playback Mode** drop-down list. To play the animation back and forth continuously, reversing the playback direction each time, select **Auto Reverse** in the **Playback Mode** drop-down list.

To turn off the continuous playback, select **Play Once** in the **Playback Mode** list. This is the default setting.

28.18.2.6. Stopping the Animation

To stop the animation during playback, click the "stop" button (the square in the middle of the playback control buttons). If your animation contains very complicated scenes, there may be a slight delay before the animation stops.

28.18.2.7. Advancing the Animation Frame by Frame

To advance the animation manually frame by frame, use the third button from the right (a vertical bar with a triangle pointing to the right). Each time you click this button, the next frame will be displayed in the graphics window. To reverse the animation frame by frame, use the third button from the left (a left-pointing triangle with a vertical bar). Frame-by-frame playback allows you to freeze the animation at points that are of particular interest.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.18.2.8. Deleting an Animation Sequence

If you want to remove one of the sequences that you have created or read in, select it in the **Sequences** list and click the **Delete** button. If you want to delete all sequences, click the **Delete All** button.

Important

Note that if you delete a sequence that has not yet been saved to disk (i.e., if you selected **In Memory** under **Storage Type** in the **Animation Sequence** dialog box), it will be removed from memory permanently. If you want to keep any animation sequences that are stored only in memory, you should be sure to save them (as described in *Saving an Animation Sequence* (p. 1302)) before you delete them from the **Sequences** list or exit ANSYS FLUENT.

28.18.3. Saving an Animation Sequence

Once you have created an animation sequence, you can save it in any of the following formats:

- Solution animation file containing the ANSYS FLUENT metafiles
- Picture files, each containing a frame of the animation sequence
- MPEG file containing each frame of the animation sequence

Note that, if you are saving picture files or an MPEG file, you can modify the view (e.g., translate, rotate, zoom) in the graphics window where the animation is displayed, and save the modified view instead of the original view.

28.18.3.1. Solution Animation File

If you selected **Metafile** or **PPM Image** under **Storage Type** in the *Animation Sequence Dialog Box* (p. 1953), then ANSYS FLUENT will save the solution animation file for you automatically. It will be saved in the specified **Storage Directory**, and its name will be the **Name** you specified for the sequence, with a `.cxa` extension (e.g., `pressure-contour.cxa`). In addition to the `.cxa` file, ANSYS FLUENT will also save a metafile with a `.hmf` extension for each frame (e.g., `pressure-contour_0002.hmf`). The `.cxa` file contains a list of the associated `.hmf` files, and tells ANSYS FLUENT the order in which to display them.

If you selected **In Memory** under **Storage Type**, then the solution animation file (`.cxa`) and the associated metafiles (`.hmf`) will be lost when you exit from ANSYS FLUENT, unless you save them as described below.

You can save the animation sequence to a file that can be read back into ANSYS FLUENT (see *Reading an Animation Sequence* (p. 1303)) when you want to replay the animation. As noted in *Reading an Animation Sequence* (p. 1303), the solution animation file can be used for playback in ANSYS FLUENT independent of the case and data files that were used to generate it.

To save a solution animation file (and the associated metafiles), select **Animation Frames** in the **Write/Record Format** drop-down list in the **Playback** dialog box, and click the **Write** button. ANSYS FLUENT will save a `.cxa` file, as well as a `.hmf` file for each frame of the animation sequence. The filename for the `.cxa` file will be the specified sequence **Name** (e.g., `pressure-contour.cxa`), and the file names for the metafiles will consist of the specified sequence **Name** followed by a frame number (e.g., `pressure-contour_0002.hmf`). All of the files (`.cxa` and `.hmf`) will be saved in the current working directory.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.18.3.2. Picture File

You can also generate a picture file for each frame in the animation sequence. This feature allows you to save your sequence frames to picture files used by an external animation program such as ImageMagick. As noted above, you can modify the view in the graphics window before you save the picture files.

To save the animation as a series of picture files, follow these steps:

1. Select **Picture Files** in the **Write/Record Format** drop-down list in the **Playback** dialog box.
2. If necessary, click the **Picture Options...** button to open the *Save Picture Dialog Box* (p. 1988) and set the appropriate parameters for saving the picture files. (If you are saving picture files for use with ImageMagick, for example, you may want to select the window dump format. See *Window Dumps (Linux/UNIX Systems Only)* (p. 121) **Apply** in the **Save Picture** dialog box to save your modified settings.

   Important

   Do not click the **Save...** button in the **Save Picture** dialog box. You will save the picture files from the **Playback** dialog box in the next step.

3. In the **Playback** dialog box, click the **Write** button. ANSYS FLUENT will replay the animation, saving each frame to a separate file. The filenames will consist of the specified sequence **Name** followed by an animation sequence and a frame number (e.g., `pressure-contour_1_0002.ps`), and they will all be saved in the current working directory.

28.18.3.3. MPEG File

It is also possible to save all of the frames of the animation sequence in an MPEG file, which can be viewed using an MPEG decoder such as **mpeg_play**. Saving the entire animation to an MPEG file will require less disk space than storing individual window dump files (using the picture method), but the MPEG file will yield lower-quality images.

As noted above, you can modify the view in the graphics window before you save the MPEG file.

To save the animation to an MPEG file, follow these steps:

1. Select **MPEG** in the **Write/Record Format** drop-down list in the **Playback** dialog box.
2. Click the **Write** button.

   ANSYS FLUENT will replay the animation and save each frame to a separate scratch file, and then it will combine all the files into a single MPEG file. The name of the MPEG file will be the specified sequence **Name** with an `.mpg` extension (e.g., `pressure-contour.mpg`), and it will be saved in the current working directory.

28.18.4. Reading an Animation Sequence

If you have saved an animation sequence to a solution animation file (as described in *Saving an Animation Sequence* (p. 1302)), you can read that file back in at a later time (or in a different session) and play the animation. Note that you can read a solution animation file into any ANSYS FLUENT session; you do not need to read in the corresponding case and data files. In fact, you do not need to read in any case and data files at all before you read a solution animation file into ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To read a solution animation file, click the **Read...** button in the *Playback Dialog Box* (p. 1990). In *The **Select File** Dialog Box* (p. 33), specify the name of the file to be read.

28.19. Checking Your Case Setup

After you have set up your case, and prior to solving it, you can check your case setup using the *Case Check Dialog Box* (p. 1959) (*Figure 28.46* (p. 1304)). This function provides you with guidance and best practices when choosing case parameters and models. Your case will be checked for compliance in the mesh, models, boundary and cell zone conditions, material properties, and solver categories. Established rules will be available for each category, with recommended changes to your current settings. At your discretion, you may elect to apply the recommendations, or keep your current settings.

To access the *Case Check Dialog Box* (p. 1959) (*Figure 28.46* (p. 1304)), go to

Run Calculation → **Check Case...**

If there are no problems with your case setup, then an information dialog box (*Figure 28.47* (p. 1304)) will appear stating that no recommendations need to be made at this time, otherwise, the *Case Check Dialog Box* (p. 1959) will open.

Figure 28.46 The Case Check Dialog Box

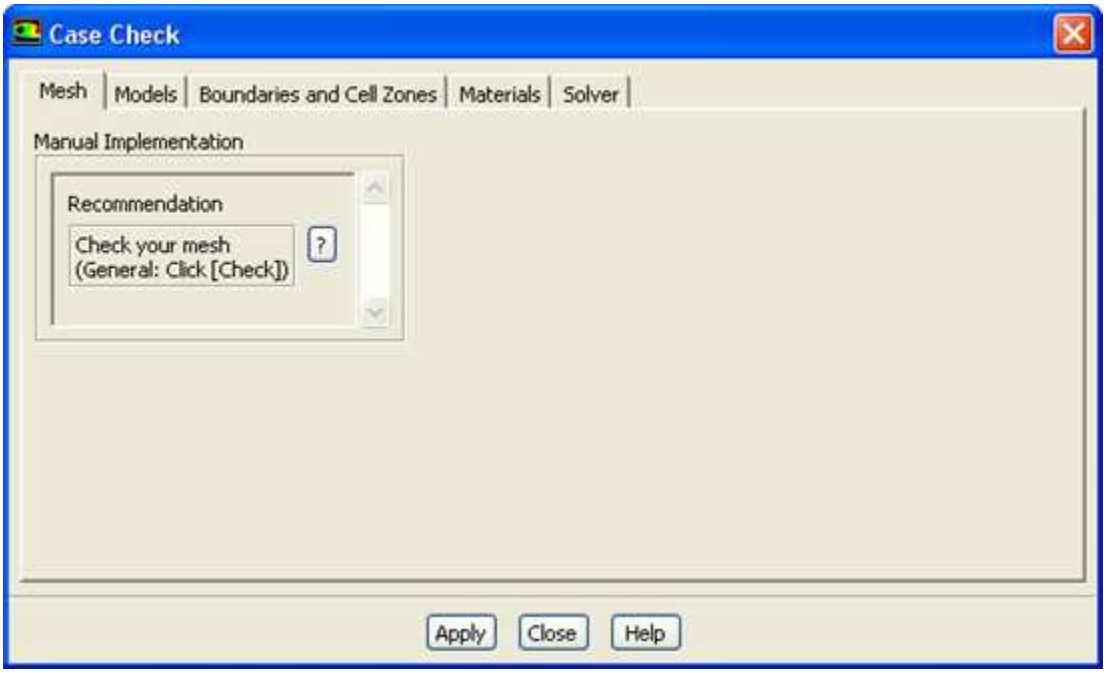
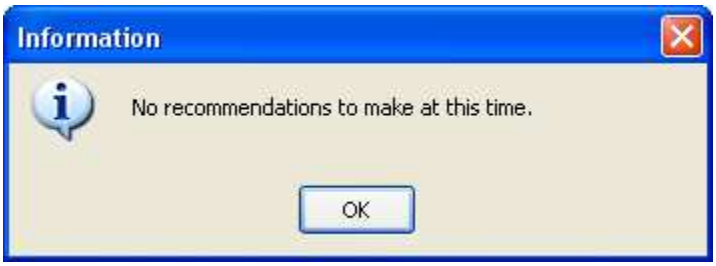


Figure 28.47 The Information Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Case Check** dialog box, each of the tabs **Mesh**, **Models**, **Boundaries and Cell Zones**, **Materials**, and **Solver** may contain recommendations. For each of the tabs that are enabled, best practices will be listed.

In some cases, the dialog box will be split based on the method that the recommendation is applied. There are two ways you can apply the listed recommendations:

Automatic Implementation
ANSYS FLUENT applies the change for you.

Manual Implementation
You will manually change your case settings.

For additional information, please see the following sections:
28.19.1. Automatic Implementation
28.19.2. Manual Implementation

28.19.1. Automatic Implementation

To the left of each of the recommendations listed under **Automatic Implementation** (e.g., *Figure 28.49* (p. 1308)), there is an enabled **Apply** check box. An enabled check box will result in ANSYS FLUENT applying the change to your case automatically. If there are some recommendations that you do not want ANSYS FLUENT to implement automatically, then click the **Apply** check box to toggle off and disable the implementation of a particular recommendation. After going through all the tabs and determining which rules you want applied automatically, click the **Apply** button at the bottom of the dialog box. Changes to your settings will be applied to all recommendations throughout the dialog box with an enabled **Apply** check box. ANSYS FLUENT will print a message in the console notifying you that the applied recommendation has been implemented.

ANSYS FLUENT will ask you if you want to save the case before proceeding to the next step. If you choose **Yes**, *The **Select File** Dialog Box* (p. 33) will open allowing you to save your case with the new settings. If you select **No**, all the changes made to the case file will be lost once you exit ANSYS FLUENT.

28.19.2. Manual Implementation

For recommendations that are listed under **Manual Implementation**, ANSYS FLUENT cannot apply the changes for you. Therefore, if you opt to make a change to your current settings, based on the listed recommendations, then you will need to manually make the changes by opening the affected dialog boxes or task pages and applying what was recommended.

To the right of the recommendations is a **?**, which essentially acts as a help button, leading you to related documentation on the specific topic.

At the bottom of each recommendation, there is a path which will guide you to the dialog box or task page where you can make the changes. For example, in the **Mesh** tab, you will see the following recommendation:

```
Check your mesh.
(General: Click [Check])
```

To perform the action, highlight **General** in the navigation pane, then click the **Check** button in the **Mesh** group box. You will see a path for each recommendation, in each of the tabs.

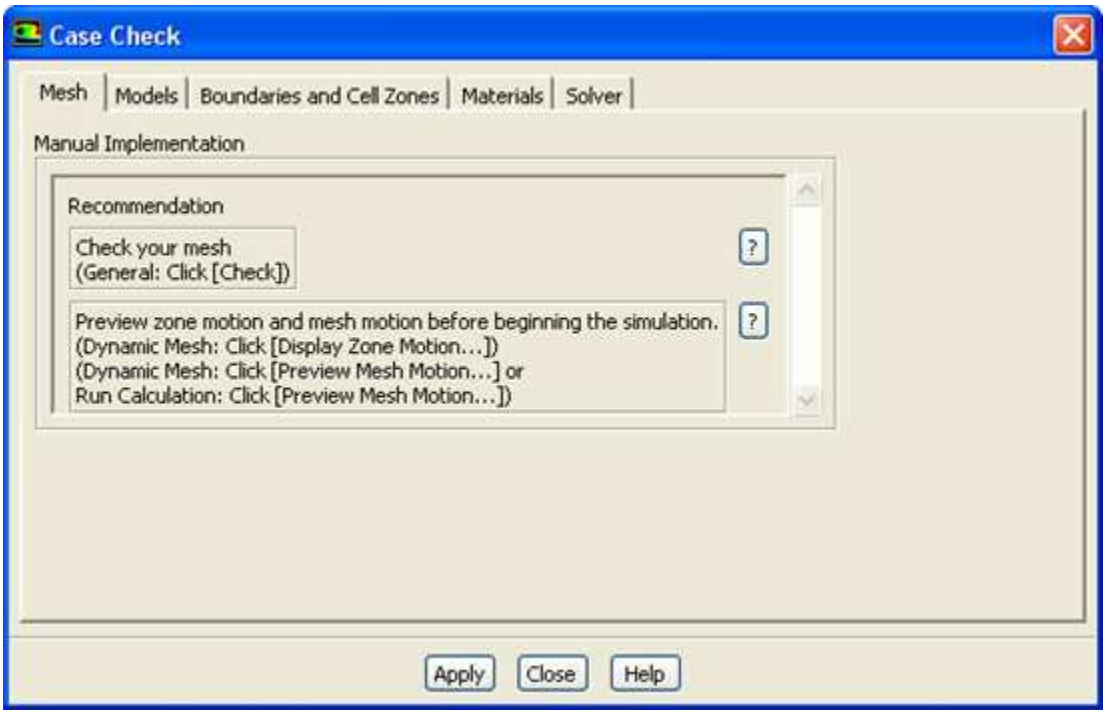
Each of the case check rules are described in the following sections:
28.19.2.1. Checking the Mesh
28.19.2.2. Checking Model Selections
28.19.2.3. Checking Boundary and Cell Zone Conditions
28.19.2.4. Checking Material Properties

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.19.2.5. Checking the Solver Settings

28.19.2.1. Checking the Mesh

Figure 28.48 The Mesh Tab in the Case Check Dialog Box



The following recommendations appear under the **Mesh** tab (*Figure 28.48* (p. 1306)):

- **Check your mesh.**

  If you have not already checked your mesh, it is best practice that you do so immediately after reading in your mesh, or after any mesh modification. To check your mesh go to

  General → **Check**

  Checking the mesh will help you detect any mesh trouble before you get started with your problem setup. You can learn more about the information obtained when checking your mesh, by going to *Checking the Mesh* (p. 172).

- **Improve the mesh quality before proceeding with your simulation. The maximum cell skewness is greater than 0.98.**

  Check the quality of your mesh immediately after reading in your mesh, or after any mesh modification. The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. You can learn more about the quality of your mesh by going to *Mesh Quality* (p. 144).

  General → **Report Quality**

- **Preview zone motion and mesh motion before beginning the simulation.**

  After setting up your case using the **Dynamic Mesh** model, it is worth while to preview your mesh prior to running your simulation. You can preview **Zone Motion** by going to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Mesh → **Dynamic Mesh** → **Display Zone Motion...**

To preview the **Mesh Motion**, go to

Dynamic Mesh → **Dynamic Mesh** → **Preview Mesh Motion...**

Run Calculation → **Preview Mesh Motion...**

It is important that you preview zone motion first and then mesh motion. Zone motion shows the motion of all dynamic zones with the prescribed rigid body motion, using the graphics library. It is a very fast process and does not alter the mesh. Previewing the zone motion will show you if the motion is setup properly (e.g. zones moving in the wrong direction or rotating about the wrong center). It is much more difficult to detect these problems with mesh motion because small time steps are performed and no continuous animation is shown.

Mesh motion should always be done for dynamic mesh cases with prescribed motion. Mesh motion will only show the validity of the mesh during the simulation. Mesh deformation and dynamic zones without rigid body motion will be considered during a mesh motion preview.

Both the **Mesh Motion** and **Zone Motion** dialog boxes will have a **Preview** button which will allow you to view the mesh or zone motion prior to running your case. You can obtain more information on mesh motion and zone motion by going to *Previewing the Dynamic Mesh* (p. 626).

- **Translate the mesh for axisymmetric geometry containing nodes below the x-axis.**

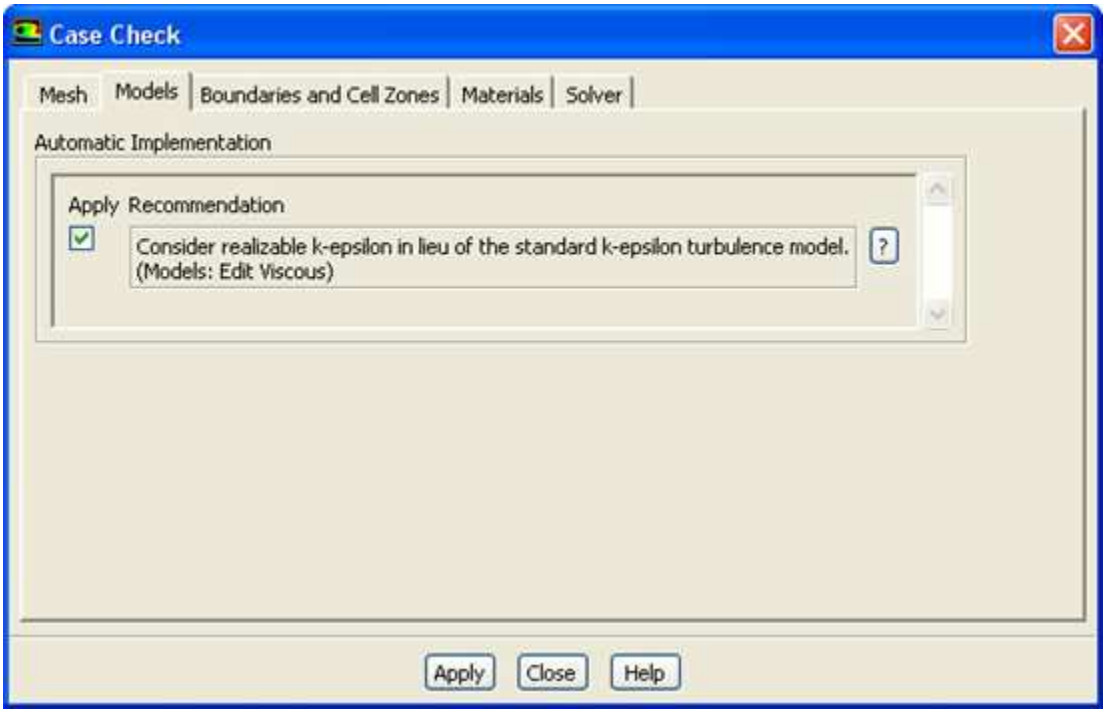
  If either **Axisymmetric** or **Axisymmetric Swirl** is specified in the **General** task page and there are mesh nodes that fall below the X-axis, then it is recommended that you translate the mesh. Nodes below the x axis are forbidden for axisymmetric cases, since the axisymmetric cell volumes are created by rotating the 2D cell volume about the x axis; thus nodes below the x axis would create negative volumes. To find out if there are any nodes that lie below the x-axis, perform a mesh check (*Checking the Mesh* (p. 172)). For information on translating the mesh, see *Translating the Mesh* (p. 202). To access the **Mesh Translate** dialog box, go to

  Mesh → **Translate...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.19.2.2. Checking Model Selections

Figure 28.49 The Models Tab in the Case Check Dialog Box



The following recommendations appear under the **Models** tab (*Figure 28.49* (p. 1308)):

- **Consider realizable k-epsilon in lieu of the standard k-epsilon turbulence model.**

  The realizable k- ε model is a more recent development of the standard k- ε model and differs from it in that the realizable k- ε model contains a new formulation for the turbulent viscosity, as well as a new transport equation for the dissipation rate, ε, derived from an exact equation for the transport of the mean-square vorticity fluctuation.

  realizable k- ε model means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. For more information on the standard k- ε model and the realizable k- ε model, visit Standard k- ε Model and Realizable k- ε Model (in the Theory Guide), respectively.

  Models → **Viscous** → **Edit...**

  For information on all k- ε model options, go to Standard, RNG, and Realizable k- ε Models in the Theory Guide.

- **Disable DO/Energy coupling if the optical thickness is less than 10.**

  DO/Energy coupling should only be used when the optical thickness is greater than 10. Refer to Energy Coupling and the DO Model in the Theory Guide for more information.

  Models → **Radiation** → **Edit...**

- **Verify that the temperature specified for boundary zones that do not participate in the view factor calculation is appropriate.**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using the S2S radiation model, make sure that you set the temperature for boundaries that do not participate in the view factor calculation to an appropriate value. In most cases the appropriate value is the ambient temperature, which by default is assumed to be 300 K. See *Specifying Boundary Zone Participation* (p. 715) for more information.

Models → **Radiation** → **Edit...**

- **Change the under-relaxation factor for the mixing plane model to 1.0.**

  If you have created a mixing plane, set the **Under-Relaxation** in the **Mixing Plane**dialog box to 1. Look under Global Parameters in *Setting Up the Mixing Plane Model* (p. 528) for information about the mixing plane under-relaxation.

  Define → **Mixing Planes...**

- **Enable the smoothing option for dynamic mesh simulations when remeshing.**

  When your case involves the use of dynamic meshes and remeshing is enabled, then it is recommended that you also perform smoothing on the mesh. For a complete discussion of smoothing and remeshing, see *Setting Dynamic Mesh Modeling Parameters* (p. 550).

  Dynamic Mesh → **Dynamic Mesh**

- **Disable species inlet diffusion for laminar flow with species transport.**

  By default, ANSYS FLUENT includes the diffusion flux of species at inlets. In some cases involving species transport and laminar flow, it is recommended that the **Inlet Diffusion** option in the **Species Model** dialog box is disabled. For example,

  – If you wish to include only the convective transport of species through the inlets of your domain.

  – If at one of the inlets, the convective flux is very small, resulting in mass loss by diffusion through the inlet.

  Models → **Species** → **Edit...**

  For more information about diffusion at inlets, go to *Defining Cell Zone and Boundary Conditions for Species* (p. 822).

- **Include turbulence interaction for the NOx model.**

  When running thermal NOx simulations and your flow is turbulent, then be sure to set the NOx **Turbulence Interaction Mode**.

  Models → **NOx** → **Edit...**

  In turbulent combustion calculations, ANSYS FLUENT solves the density-weighted time-averaged Navier-Stokes equations for temperature, velocity, and species concentrations or mean mixture fraction and variance. Methods of modeling the mean turbulent reaction rate can be based on either moment methods or probability density function (PDF) techniques. ANSYS FLUENT uses the PDF approach.

  To learn about how this feature is set up, go to *Setting Turbulence Parameters* (p. 955).

- **Consider using the default Schnerr-Sauer or the Zwart-Gerber-Belamri cavitation model.**

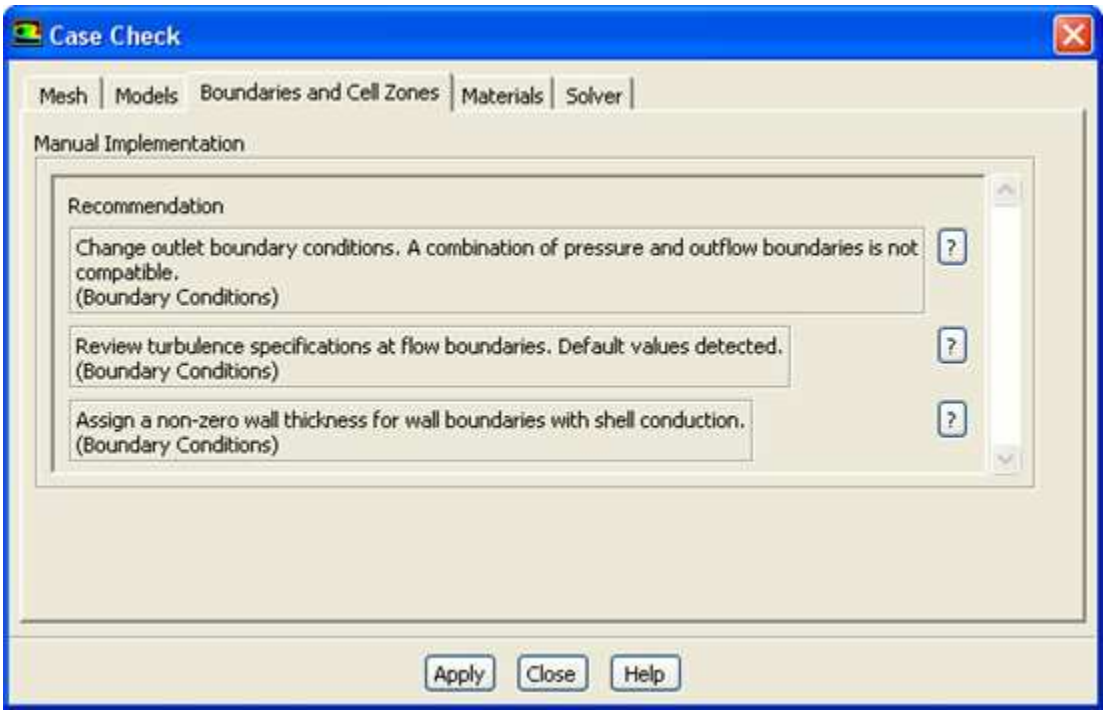
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using the mixture multiphase model with the Singhal et. al cavitation model enabled, consider changing it to either the Schnerr-Sauer or the Zwart-Gerber-Belamri cavitation model. Refer to Cavitation Models in the Theory Guide for more information.

Phases → Interaction...

28.19.2.3. Checking Boundary and Cell Zone Conditions

Figure 28.50 The Boundaries and Cell Zones Tab in the Case Check Dialog Box



The following recommendations appear under the **Boundaries and Cell Zones** tab (*Figure 28.50* (p. 1310)):

- **Apply an axis boundary on the centerline (x-axis).**

  For geometry that is axisymmetric or axisymmetric swirl (as set in the **General** task page), the centerline (x-axis) boundary type should be set to **axis**. See *Axis Boundary Conditions* (p. 323).

  Boundary Conditions

- **Change inlet boundary conditions. Velocity inlet boundary conditions are not compatible with compressible flow.**

  This boundary condition is intended for incompressible flows, and its use in compressible flows will lead to a nonphysical result because it allows stagnation conditions to float to any level (see *Velocity Inlet Boundary Conditions* (p. 264)). If you decide to select a different boundary type, go to the **Boundary Conditions** task page.

  Boundary Conditions

- **Change outlet boundary conditions. A combination of pressure and outflow boundaries is not compatible.**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Outflow boundary conditions in ANSYS FLUENT are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. One of the limitations when using outflow boundary conditions is that outflow boundary conditions are not compatible with pressure inlets. Therefore, it is recommended that you use velocity or mass flow inlets instead of pressure inlets when used in combination with outflow boundaries. See *Outflow Boundary Conditions* (p. 293) for a list of limitations that exist with outflow boundaries.

Boundary Conditions

- **Change outlet boundary conditions. Outflow boundary conditions are not compatible with the ideal gas law for density.**

  Outflow boundaries cannot be used if you are modeling unsteady flows with varying density, even if the flow is incompressible. See *Outflow Boundary Conditions* (p. 293) for more limitations that exist with outflow boundaries.

Boundary Conditions

- **Non-zero operating pressure set. This will be added to gauge pressure inputs.**

  For cases that have density specified as the ideal gas law, and the operating pressure is greater than zero, the operating pressure will be added to the gauge pressure to yield the absolute pressure. For more information, see *Density Inputs for the Ideal Gas Law for Compressible Flows* (p. 406) and *Operating Pressure, Gauge Pressure, and Absolute Pressure* (p. 449).

Boundary Conditions → **Operating Conditions...**

- **Apply positive non-zero pressure boundary conditions when using the ideal gas law for density.**

  In compressible flows , isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, p'_0, at the inlet and the static pressure, p'_s, in the adjacent fluid cell are related, as described in *Equation 7–62* (p. 264) *Equation 7–63* (p. 264) of *Calculation Procedure at Pressure Inlet Boundaries* (p. 263). It is recommended that pressure boundary conditions are not set to zero for compressible flows that use the ideal gas law.

Boundary Conditions

- **Review turbulence specifications at flow boundaries. Default values detected.**

  If your case setup has any of the turbulence models enabled, be sure to review the default parameters for the **K and Epsilon Turbulence Specification Method** in the outlet and inlet boundary conditions. ANSYS FLUENT's default parameters for the **Backflow Turbulent Kinetic Energy** and **Backflow Turbulent Dissipation Rate** are 1. You can either adjust the values, or select a different **Turbulence Specification Method**. For general information turbulence parameters, see *Determining Turbulence Parameters* (p. 250).

Boundary Conditions

- **Assign a non-zero wall thickness for wall boundaries with shell conduction.**

  When the **Shell Conduction** option is enabled in the **Wall** boundary condition dialog box, ANSYS FLUENT will compute heat conduction within the wall, in addition to conduction across the wall. therefore, you must specify a non-zero **Wall Thickness** in the **Wall** dialog box, because the shell con-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

duction model is relevant only for walls with non-zero thickness. See *Shell Conduction in Thin-Walls* (p. 313) for information on shell conduction in thin walls.

Boundary Conditions

- **Assign a value of 0 or 1 for VOF at the inlet or outlet boundary conditions.**

  When enabling the VOF model, the **Volume Fraction** in the inlet and outlet boundary conditions for each phase should be set either to `0` or `1`. No intermediate values are permitted. For general information on boundary condition setup, see *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107).

Boundary Conditions

- **Change the outlet boundary condition. Outflow boundary condition is not compatible with current multiphase settings.**

  You cannot assign an outflow boundary condition when using the mixture and Eulerian multiphase models. Note the limitations of this boundary condition in *Outflow Boundary Conditions* (p. 293). ANSYS FLUENT can model the effects of open channel flow using the VOF formulation. In such a case, outflow boundary conditions can be used at the outlet of open channel flows, to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. See Open Channel Flow in the Theory Guide, under the heading Outflow Boundary, for more information.

Boundary Conditions

- **Review wall motion. Stationary wall motion relative to adjacent cell zone detected.**

  In cases where the fluid zone motion type is specified as **Moving Mesh**or **Moving Reference Frame**, all wall zones should be set to **Moving Wall** in the **Momentum** tab in the **Wall**boundary conditions dialog box. The wall motion should be defined **Relative to Adjacent Cell Zone**. The exception to this is if the walls are stationary in the absolute frame. To define wall motion, see *Inputs at Wall Boundaries* (p. 300).

Boundary Conditions

- **Assign non-zero velocities when specifying a moving fluid zone.**

  If selecting either **Moving Mesh** or **Moving Reference Frame** in the **Fluid** dialog box, be sure to set non-zero values for the rotational and translational velocities. Refer to *Defining Zone Motion* (p. 217) for user inputs.

Cell Zone Conditions

- **Review flow specifications at inlet boundaries. Default values detected.**

  For **mass-flow-inlet** and **velocity-inlet** boundary conditions, the default values in ANSYS FLUENT are $1kg/s$ and $0m/s$, respectively. Review the settings and adjust accordingly. See *Default Settings at Velocity Inlet Boundaries* (p. 269) and *Default Settings at Mass Flow Inlet Boundaries* (p. 276) for default parameters of velocity inlets and mass flow inlets, respectively.

Boundary Conditions

- **Define the porous zone when using the heat exchanger model.**

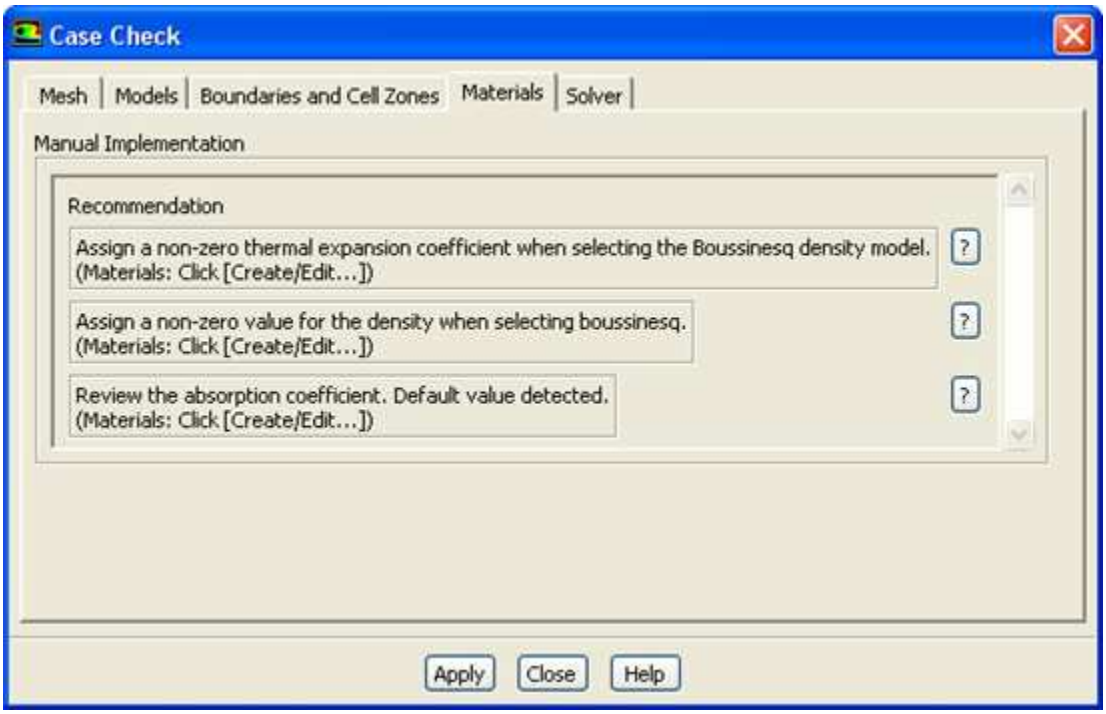
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat exchanger models always require the definition of the porous media zone on the primary side for the macro model and for both primary and auxiliary sides for the dual cell model. See Streamwise Pressure Drop in the Theory Guide for more information.

Cell Zone Conditions

28.19.2.4. Checking Material Properties

Figure 28.51 The Materials Tab in the Case Check Dialog Box



The following recommendations appear under the **Materials** tab (*Figure 28.51* (p. 1313)):

- **Assign individual fluid Cps to polynomial functions of temperature.**

  For cases with species transport and volumetric reactions, it is best practice to specify the specific heat capacity C_p as a polynomial that is a function of temperature. See *Defining Properties for the Mixture and Its Constituent Species* (p. 805) and *Specific Heat Capacity as a Function of Temperature* (p. 434) for information on defining material properties for the species in the mixture.

  Materials

- **Assign a non-zero value for the density when selecting boussinesq.**

  The Boussinesq model is used for natural convection problems involving small changes in temperature. To enable the Boussinesq approximation for density, choose **boussinesq** from the **Density** drop-down list in the **Create/Edit Materials** dialog box and specify a constant value for **Density**. See *Inputs for the Boussinesq Approximation* (p. 404).

  Materials

- **Review the absorption coefficient. Default value detected.**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If any of the radiation models are enabled. Enter an absorption coefficient for the material listed (*Radiation Properties* (p. 435)).

Materials

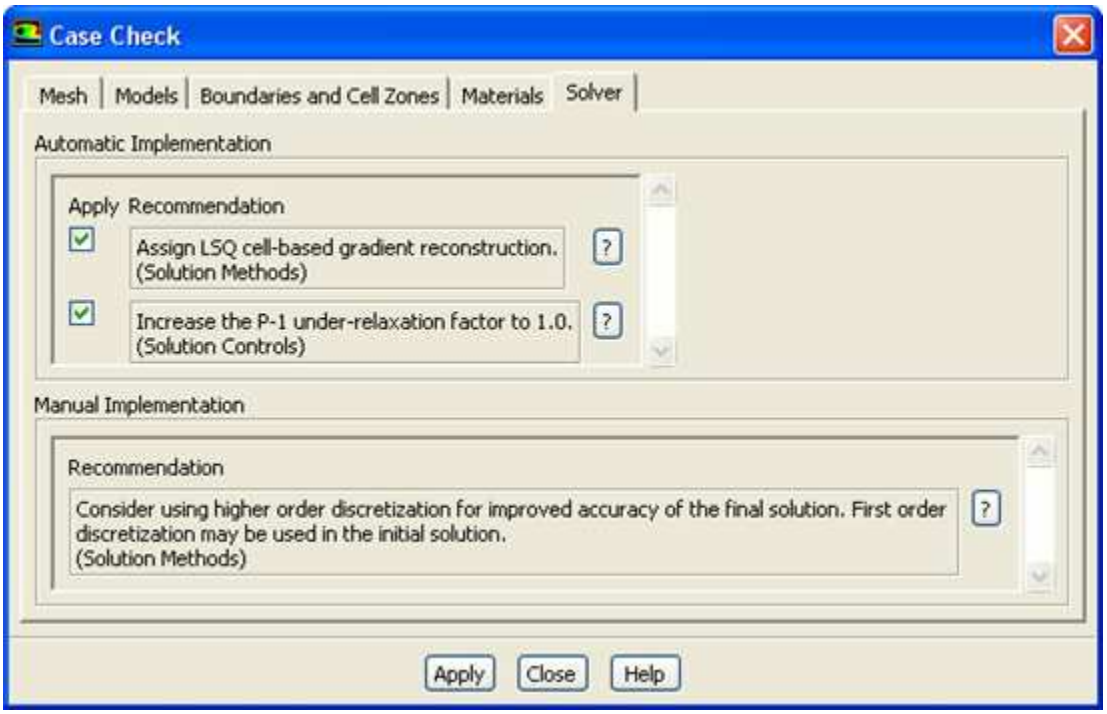
- **Assign a non-zero thermal expansion coefficient when selecting the Boussinesq density model.**

  When selecting **boussinesq** to describe the density of your material, be sure to enter a valid thermal expansion coefficient for your material. For detailed information on the Boussinesq model, see *The Boussinesq Model* (p. 697).

Materials

28.19.2.5. Checking the Solver Settings

Figure 28.52 The Solver Tab in the Case Check Dialog Box



The following recommendations appear under the **Solver** tab (*Figure 28.52* (p. 1314)):

- **Enable the unsteady solver option when selecting moving mesh for the fluid boundary.**

  If the motion type of the fluid boundary condition is specified as **Moving Mesh**, then your case should be specified as **Transient** in the **General** task page. Visit *Setting Up the Sliding Mesh Problem* (p. 541) for steps on setting up moving mesh problem.

General

- **Assign LSQ cell-based gradient reconstruction.**

  The least squares cell-based averaging scheme is known to be as accurate as the node-based gradient for irregular unstructured meshes, but less expensive to compute than the node-based gradient.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, it is recommended that least squares cell-based gradient reconstruction is used. See Evaluation of Gradients and Derivatives in the Theory Guide for more information on gradient options.

Solution Methods

- **Change the under-relaxation factor for the energy equation to at least 0.90.**

  It is recommended to set the energy under-relaxation factor between 0.90 and 1.0. If you decide to apply this recommendation, then ANSYS FLUENT will automatically set the energy under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See *Solution Strategies for Heat Transfer Modeling* (p. 693) for the underrelaxation of the energy equation.

Solution Controls

- **Review reference values for non-dimensionalized force coefficients computed with force monitors.**

  If force monitors are activated, be sure to update the **Reference Values** task page. See *Monitoring Force and Moment Coefficients* (p. 1278).

Solution Controls

- **Increase the NOx under-relaxation factor to at least 0.90.**

  If the NOx model is enabled, set the NOx under-relaxation factor to a value of at least 0.90 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then ANSYS FLUENT will automatically set the NOx under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See *Using the NOx Model* (p. 943).

Solution Controls

- **Increase the Discrete Ordinates under-relaxation factor to at least 0.90.**

  If the Discrete Ordinates (DO) radiation model is enabled, set the radiation under-relaxation factor to a value of at least 0.90 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then ANSYS FLUENT will automatically set the radiation under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See *DO Solution Parameters* (p. 735).

Solution Controls

- **Increase the P1 under-relaxation factor to 1.0.**

  If the P1 radiation model is enabled, set the radiation under-relaxation factor to 1.0 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then ANSYS FLUENT will automatically set the radiation under-relaxation factor to `1.0`. See *P-1 Model Solution Parameters* (p. 733).

Solution Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Increase the species and energy under-relaxation factors to at least 0.90.**

  For a case with species transport and energy defined, set the species and energy under-relaxation factors to a value of at least 0.90. If you decide to apply this recommendation, then ANSYS FLUENT will automatically set the species and energy under-relaxation factors to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See *Solution Procedures for Chemical Mixing and Finite-Rate Chemistry* (p. 823).

  Solution Controls

- **Assign a value of 1 for the under-relaxation factor for unsteady DPM with 1 DPM update per time step.**

  It is recommended that the DPM under-relaxation factor be set to 1 for unsteady DPM with 1 DPM update per time step.

  Solution Controls

- **Increase the mean mixture fraction under-relaxation factor to at least 0.90.**

  If the non-premixed or partially premixed combustion models are enabled, then it is best to set the mean mixture under-relaxation factor to a value of at least 0.90 to ensure full convergence. If you decide to apply this recommendation, then ANSYS FLUENT will automatically set the mean mixture under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See *Solving the Flow Problem* (p. 887).

  Solution Controls

- **Consider using higher order discretization for improved accuracy of the final solution. First-order discretization may be used in the initial solution.**

  It is generally advisable to obtain an initial solution using first-order accurate discretization, however, second order discretization is recommended for improved accuracy of the final solution. See *Choosing the Spatial Discretization Scheme* (p. 1210) for more information on discretization schemes.

  Solution Methods

- **Select the absolute reference frame for initializing cases when using the MRF model.**

  When using the MRF model, always use the absolute reference frame while initializing the solution. Select **Absolute** under **Reference Frame** in the **Solution Initialization** task page. If the **Relative to Cell Zone** option is selected, which is the default option, the initial flow field can contain discontinuities, which can cause convergence problems in the first few iterations. Refer to *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240) for more information.

  Solution Initialization → Initialize

- **Choose PRESTO! for the pressure discretization scheme.**

  When using the VOF model, it is recommended that you use **PRESTO!** as the pressure discretization scheme. This scheme is recommended for flows with high swirl numbers, a high-Rayleigh-number natural convection, high-speed rotating flows, flows involving porous media, and flows in strongly curved domains. See *Choosing the Pressure Interpolation Scheme* (p. 1212) for more information.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Methods

28.20. Convergence and Stability

Convergence can be hindered by a number of factors. Large numbers of computational cells, overly conservative under-relaxation factors, and complex flow physics are often the main causes. Sometimes it is difficult to know whether you have a converged solution. In the following sections, some of the numerical controls and modeling techniques that can be exercised to enhance convergence and maintain stability are examined.

28.20.1. Judging Convergence
28.20.2. Step-by-Step Solution Processes
28.20.3. Modifying Algebraic Multigrid Parameters
28.20.4. Modifying the Multi-Stage Parameters

You should also refer to *Choosing the Spatial Discretization Scheme* (p. 1210) and *Choosing the Pressure-Velocity Coupling Method* (p. 1215) for information about how the choice of discretization scheme or (for the pressure-based solver) pressure-velocity coupling scheme can affect convergence. Manipulation of under-relaxation parameters and multigrid settings to enhance convergence is discussed in *Setting Under-Relaxation Factors* (p. 1217) and *Modifying Algebraic Multigrid Parameters* (p. 1319).

28.20.1. Judging Convergence

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities such as drag or heat transfer coefficient.

For most problems, the default convergence criterion in ANSYS FLUENT is sufficient. This criterion requires that the globally scaled residuals, defined by *Equation 28–5* (p. 1269) or *Equation 28–11* (p. 1271) decrease to 10^{-3} for all equations except the energy and P-1 equations, for which the criterion is 10^{-6}. Locally scaled residuals defined by *Equation 28–6* (p. 1270) or *Equation 28–12* (p. 1271) decrease to 10^{-5} for all equations.

Sometimes, however, this criterion may not be appropriate. Typical situations are listed below.

- If you make a good initial guess of the flow field, the initial continuity residual may be very small leading to a large scaled residual for the continuity equation. In such a situation it is useful to examine the unscaled residual and compare it with an appropriate scale, such as the mass flow rate at the inlet.
- For some equations, such as for turbulence quantities, a poor initial guess may result in high scale factors. In such cases, scaled residuals will start low, increase as non-linear sources build up, and eventually decrease. It is therefore good practice to judge convergence not just from the value of the residual itself, but from its behavior. You should ensure that the residual continues to decrease (or remain low) for several iterations (say 50 or more) before concluding that the solution has converged.

Another popular approach to judging convergence is to require that the unscaled residuals drop by three orders of magnitude. ANSYS FLUENT provides residual normalization for this purpose, as discussed in *Definition of Residuals for the Pressure-Based Solver* (p. 1269), where residuals are defined for both the pressure-based solver and the density-based solver. In this approach the convergence criterion is that the normalized unscaled residuals should drop to 10^{-3}. However, this requirement may not be appropriate in many cases:

- If you have provided a very good initial guess, the residuals may not drop three orders of magnitude. In a nearly-isothermal flow, for example, energy residuals may not drop three orders if the initial guess of temperature is very close to the final solution.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the governing equation contains non-linear source terms which are zero at the beginning of the calculation and build up slowly during computation, the residuals may not drop three orders of magnitude. In the case of natural convection in an enclosure, for example, initial momentum residuals may be very close to zero because the initial uniform temperature guess does not generate buoyancy. In such a case, the initial nearly-zero residual is not a good scale for the residual.
- If the variable of interest is nearly zero everywhere, the residuals may not drop three orders of magnitude. In fully-developed flow in a pipe, for example, the cross-sectional velocities are zero. If these velocities have been initialized to zero, initial (and final) residuals are both close to zero, and a three-order drop cannot be expected.

In such cases, it is wise to monitor integrated quantities, such as drag or overall heat transfer coefficient, before concluding that the solution has converged. It may also be useful to examine the un-normalized unscaled residual, and determine if the residual is small compared to some appropriate scale. Alternatively, the scaled residual defined by *Equation 28–5* (p. 1269) or *Equation 28–11* (p. 1271) (the default) may be considered.

Conversely, it is possible that if the initial guess is very bad, the initial residuals are so large that a three-order drop in residual does not guarantee convergence. This is specially true for k and ε equations where good initial guesses are difficult. Here again it is useful to examine overall integrated quantities that you are particularly interested in. If the solution is unconverged, you may drop the convergence tolerance, as described in *Modifying Convergence Criteria* (p. 1276).

28.20.2. Step-by-Step Solution Processes

One important technique for speeding convergence for complex problems is to tackle the problem one step at a time. When modeling a problem with heat transfer, you can begin with the calculation of the isothermal flow. To solve turbulent flow, you might start with the calculation of laminar flow. When modeling a reacting flow, you can begin by computing a partially converged solution to the non-reacting flow, possibly including the species mixing. When modeling a discrete phase, such as fuel evaporating from droplets, it is a good idea to solve the gas-phase flow field first. Such solutions generally serve as a good starting point for the calculation of the more complex problems. These step-by-step techniques involve using the *Solution Controls Task Page* (p. 1902) to turn equations on and off in the **Equations** dialog box.

28.20.2.1. Selecting a Subset of the Solution Equations

ANSYS FLUENT automatically solves each equation that is turned on using the **Models** family of dialog boxes. If you specify in the *Viscous Model Dialog Box* (p. 1645) that the flow is turbulent, equations for conservation of turbulence quantities are turned on. If you specify in the *Energy Dialog Box* (p. 1645) that ANSYS FLUENT should enable energy, the energy equation is activated. Convergence can be sped up by focusing the computational effort on the equations of primary importance. The **Equations** list in the *Equations Dialog Box* (p. 1904) allows you to turn individual equations on or off temporarily.

Solution Controls → **Equations...**

A typical example is the computation of a flow with heat transfer. Initially, you will define the full problem scope, including the thermal boundary conditions and temperature-dependent flow properties. Following the problem setup, you will use the **Equations** dialog box to temporarily turn off the energy equation. You can then compute an isothermal flow field, remembering to set a reasonable initial value for the temperature of the fluid.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

This is possible only for the pressure-based solver; the density-based solver solves the energy equation together with the flow equations in a coupled manner, so you cannot turn off the energy equation as described above.

When the isothermal flow is reasonably well converged, you can turn the energy equation back on. You can actually turn off the momentum and continuity equations while the initial energy field is being computed. When the energy field begins to converge well, you can turn the momentum and continuity equations back on so that the flow pattern can adjust to the new temperature field. The temperature will couple back into the flow solution by its impact on fluid properties such as density and viscosity. The temperature field will have no effect on the flow field if the fluid properties (e.g., density, viscosity) do not vary with temperature. In such cases, you can compute the energy field without turning the flow equations back on again.

Important

If you have specified temperature-dependent flow properties, you should be sure that a realistic value has been set for temperature throughout the domain before disabling calculation of the energy equation. If an unrealistic temperature value is used, the flow properties dependent on temperature will also be unrealistic, and the flow field will be adversely affected. Instructions for initializing the temperature field or patching a temperature field onto an existing solution are provided in *Initializing the Solution* (p. 1239).

28.20.2.2. Turning Reactions On and Off

To solve a species mixing problem prior to solving a reacting flow, you should set up the problem including all of the reaction information, and save the complete case file. To turn off the reaction so that only the species mixing problem can be solved, you can use the *Species Model Dialog Box* (p. 1681) to turn off the **Volumetric** option under **Reactions**.

Models → **Species** → **Edit...**

Once the species mixing problem has partially converged, you can return to the **Species Model** dialog box and turn the **Volumetric Reactions** option on again. You can then resume the calculation starting from the partially converged data.

For combustion problems you may want to patch a hot temperature in the vicinity of the anticipated reactions before you restart the calculation. See *Patching Values in Selected Cells* (p. 1242) for information about patching an initial value for a flow variable.

28.20.3. Modifying Algebraic Multigrid Parameters

The default algebraic multigrid settings are appropriate for nearly all problems, but in rare cases you may need to make minor adjustments. *Setting Algebraic Multigrid Parameters* (p. 1226) describes how to analyze the multigrid solver's performance to determine which parameters.

28.20.4. Modifying the Multi-Stage Parameters

It is possible to make several changes to the multi-stage time-stepping scheme itself. See *Changing the Multi-Stage Scheme* (p. 1237) for detailed information.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.21. Solution Steering

For additional information, please see the following sections:

28.21.1. Overview of Solution Steering
28.21.2. Solution Steering Strategy
28.21.3. Using Solution Steering

28.21.1. Overview of Solution Steering

Solution steering in the density-based implicit solver provides you with an expert system that will help navigate the flow solution from a starting initial guess to a converged solution with minimum user interaction. When you apply solution steering, you will be required to select the type of flow that best characterizes the solution domain and the maximum desired accuracy, and then allow the solver to take the solution to convergence. As the solver proceeds with the solution iteration, certain solver parameters will be adjusted behind the scenes to insure that a converged solution to steady state is possible.

Important

Solution steering is available only for steady-state flows in the density-based implicit solver.

28.21.2. Solution Steering Strategy

The convergence to steady-state solution is achieved in two stages. The parameters that are used in these stages are determined and set based on user input for the type of flow that can best characterize the solution domain. The type of flows available for selection are classified based on flow compressibility as well as the dominant flow Mach number in the solution domain.

The following flow types are available:

- Incompressible (if the flow is incompressible, i.e. density is constant)
- Subsonic (if the flow is compressible and $M<0.75$)
- Transonic (if the flow is compressible and $0.65<M<1.2$)
- Supersonic (if the flow is compressible and $1.10< M<2.5$)
- Hypersonic (if the flow is compressible and $2.0< M$)

Important

There is no exact Mach number cut-off for these regions, therefore, the above Mach number ranges are just a simple guideline to help you select a flow type.

Solution steering will typically perform full multigrid (FMG) initialization followed by two iterative stages. The purpose of each stage is described below.

28.21.2.1. Initialization

Immediately before the start of the iteration, solution steering will perform full multigrid initialization to obtain the best possible initial starting solution.

Stage 1:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The purpose of Stage 1 is to navigate the solution from the difficult initial phase of the solution toward convergence by insuring maximum stability. During this stage, the solution is advanced gradually from 1st-order accuracy to maximum accuracy (user specified and typically 2nd-order) at a constant low CFL value.

Stage 2:

In this stage the solution is driven hard towards convergence by regular adjustments of the CFL value to insure fast convergence as well as to prevent possible divergence.

In stage 2, the residual history is monitored and analyzed through regular intervals to determine if an increase or decrease in CFL value is needed to obtain fast convergence or to prevent divergence.

28.21.3. Using Solution Steering

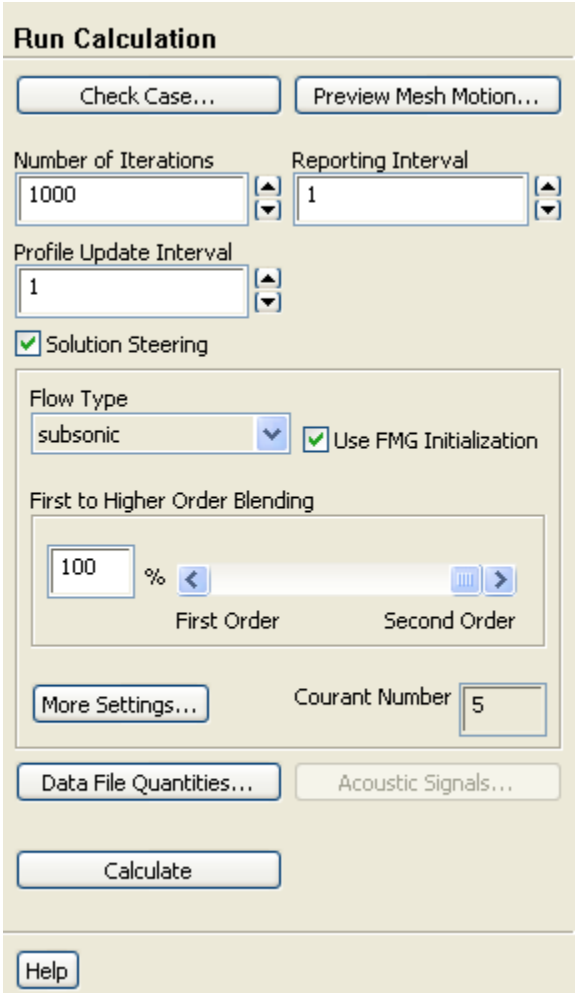
Solution steering is disabled by default. However, when the following criteria are met, the solution steering feature will become available for selection:

Feature	**Setting**
Solver Type	Density Based
Solver Formulation	Implicit
Time Formulation	Steady
Data is valid (either data file has been read or flow has been initialized)	

To activate solution steering, click the **Solution Steering** check box as shown in *Figure 28.53* (p. 1322).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.53 The Run Calculation Task Page with Solution Steering Enabled



The **Run Calculation** task page will then expand to display the solution steering main controls (see *Figure 28.53* (p. 1322)). To obtain a flow solution using solution steering, you will need to perform the following:

1. Select the type of flow.
2. Select the maximum accuracy desired (first to second order blending).
3. Click **Calculate**.

The user can also adjust the number of iterations, or customize the parameters of the solution steering if the default setting is not sufficient for the type of flow problem being solved.

Before using solution steering, you will need to prepare and set up the case as usual as described in the Getting Started Manual.

Once **Solution Steering** is activated, specify the following:

Flow Type
allows you to select the flow type that best describes the flow in the solution domain. Five choices are available: **incompressible**, **subsonic**, **transonic**, **supersonic**, and **hypersonic**.

FMG Initialization
when enabled allows for full multigrid initialization before starting stages 1 and 2. FMG initialization is enabled by default.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

First to Higher Order Blending

allows you to reduce the desired solution accuracy by selecting a blending factor less than 100%. The default setting is 100%. See First-to-Higher Order Blending in the Theory Guide for more information. The blending factor will be grayed out if **Second Order Upwind** discretization for the **Flow** equations is not selected in the **Solution Methods** task page. The solution accuracy may be reduced (typical values are 75% or 50%) if it is not possible to obtain a converged solution with the maximum second-order accuracy (i.e. blending = 100%)

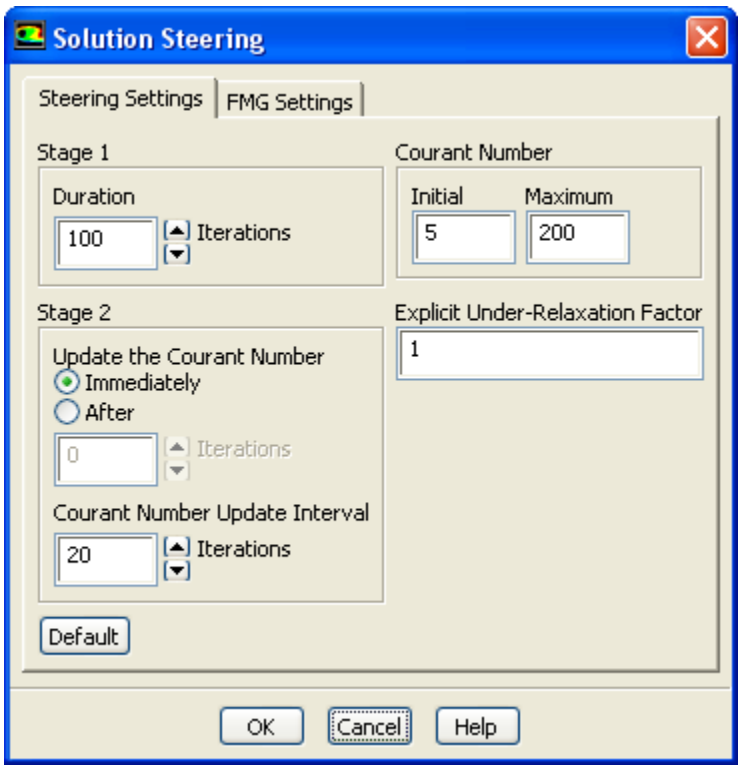
Courant Number

in the **Run Calculation** task page isa non-adjustable field displaying the current CFL number, which allows you to view it during the calculation.

More Settings...

opens the **Solution Steering** dialog box, providing a host of settings that control the solution steering strategy, as shown in *Figure 28.54* (p. 1323).

Figure 28.54 The Solution Steering Dialog Box



The **Solution Steering** dialog box, shown in *Figure 28.54* (p. 1323), contains two tabs. The **Steering Settings** tab sets the solution steering parameters and the **FMG Settings** sets the full multigrid initialization parameters.

In the **Steering Setting** tab, you can modify the parameters used in Stages 1 and 2.

Stage 1

Duration is the number of iterations in stage 1. The CFL number used during these iterations is set in the **Initial** field, in the **Courant Number** group box.

Stage 2

The Courant number update in stage 2 can start immediately after the end of stage 1, or after a certain designated number of iterations. If the Courant number update is to start immediately after stage 1 then **Immediately** should be selected (this is the default option). If the Courant number update is desired after some lagged period of iterations, then **After** should be selected and the lag in the number of iter-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ations should be entered in the field below it. The frequency at which the Courant number is updated is defined in **Courant Number Update Interval** field.

Courant Number

Initial is the starting Courant number and **Maximum**is the maximum allowed Courant number. The solution steering algorithm will not allow the solver to exceed the maximum Courant number, but will allow the solver to use a Courant number less than the initial Courant number if divergence in the solution has occurred.

Explicit Under-Relaxation Factor

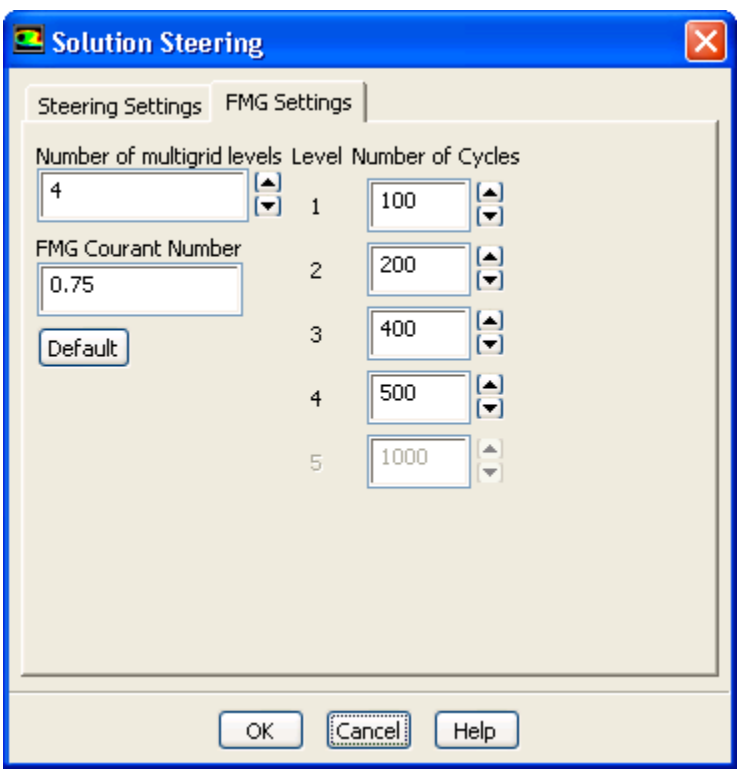
allows the solution to be under-relaxed to improve convergence. The under-relaxation value is determined by the **Flow Type** that you selected in the **Run Calculation** task page, when **Solution Steering** was enabled. In general, you do not need to alter the default value set in this field. Refer to Under-Relaxation of Variables in the Theory Guide for more information about explicit relaxation.

Default

is available in the **Steering Settings** tab to reset any changes made to the parameters to their original default values.

In the **FMG Settings** tab (*Figure 28.55* (p. 1324)), the **Number of multigrid levels** and **Number of Cycles** in each **Level**, as well as the **FMG Courant Number** used in the FMG initialization can be adjusted. The default values used in the multigrid settings are determined from the type of flow that you selected, the size of the mesh, and the flow dimensionality. The **Default** button is used to reset any changes to the original default values. For more information about FMG initialization, refer to *Full Multigrid (FMG) Initialization* (p. 1244).

Figure 28.55 The FMG Settings Tab in the Solution Steering Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 29: Adapting the Mesh

The solution-adaptive mesh refinement feature of ANSYS FLUENT allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, ANSYS FLUENT provides tools for creating and viewing adaption fields customized to particular applications. For information about the theory behind mesh adaption in ANSYS FLUENT, see "Adapting the Mesh" in the Theory Guide. Information about using the adaption process in ANSYS FLUENT is described in detail in the following sections.

29.1. Using Adaption
29.2. Boundary Adaption
29.3. Gradient Adaption
29.4. Dynamic Gradient Adaption
29.5. Isovalue Adaption
29.6. Region Adaption
29.7. Volume Adaption
29.8. Yplus/Ystar Adaption
29.9. Anisotropic Adaption
29.10. Geometry-Based Adaption
29.11. Registers
29.12. Mesh Adaption Controls
29.13. Improving the Mesh by Smoothing and Swapping

29.1. Using Adaption

Two significant advantages of the unstructured mesh capability in ANSYS FLUENT are:

- reduced setup time compared to structured meshes
- the ability to incorporate solution-adaptive refinement of the mesh

By using solution-adaptive refinement, you can add cells where they are needed in the mesh, thus enabling the features of the flow field to be better resolved. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells need to be added. Thus, computational resources are not wasted by the inclusion of unnecessary cells, as occurs in the structured mesh approach. Also, the effect of mesh refinement on the solution can be studied without completely re-generating the mesh.

Note

When you perform mesh adaption in a parallel computation, a load balancing step will be performed by ANSYS FLUENT by default.

The automatic load balancing will not occur in conjunction with dynamic adaption. See *Dynamic Gradient Adaption* (p. 1333) for information on dynamic adaption, and *Load Balancing* (p. 1614) for information on load balancing in parallel ANSYS FLUENT. For information about the static adaption process, see Static Adaption Process in the Theory Guide.

For additional information, please see the following sections:

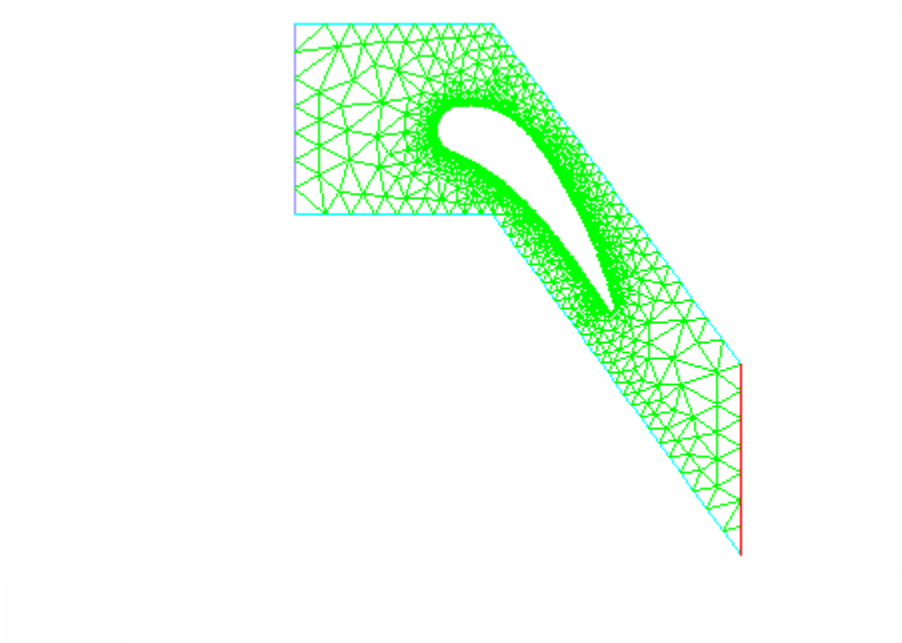
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.1.1. Adaption Example
29.1.2. Adaption Guidelines

29.1.1. Adaption Example

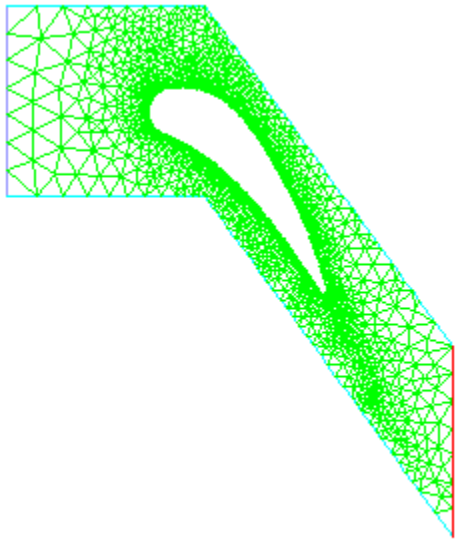
An example of the effective use of adaption is in the solution of the compressible, turbulent flow through a 2D turbine cascade. The initial mesh around the blade is fine, as shown in *Figure 29.1* (p. 1326). The surface node distribution thus provides adequate definition of the blade geometry, and enables the turbulent boundary layer to be properly resolved without further adaption. On the other hand, the mesh on the inlet, outlet, and periodic boundaries is comparatively coarse. To ensure that the flow in the blade passage is appropriately resolved, solution-adaptive refinement was used to create the mesh shown in *Figure 29.2* (p. 1327).

Figure 29.1 Turbine Cascade Mesh Before Adaption



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.2 Turbine Cascade Mesh after Adaption



While the procedure for solution adaption will vary according to the flow being solved, it is instructive to examine the adaption process used for the turbine cascade shown in the previous figure. Though this example involves compressible flow, the general procedure is applicable for incompressible flows as well.

1. Display contours of pressure adaption function to determine a suitable refinement threshold (see *Gradient Adaption* (p. 1331)).
2. "Mark the cells within the refinement threshold, creating a refinement register (see Adaption Registers in the Theory Guide and *Gradient Adaption* (p. 1331)).
3. Repeat the process described in steps 1 and 2, using gradients of Mach number as a refinement criterion.
4. To refine in the wake region, use isovalues of total pressure as a criterion (see *Isovalue Adaption* (p. 1335)). This causes cells within the boundary layer and the wake to be marked, since these are both regions of high total-pressure loss.
5. Use the *Manage Adaption Registers Dialog Box* (p. 2129) to combine the three refinement registers into a single register (see *Manipulating Adaption Registers* (p. 1343)).
6. Limit the minimum cell volume for adaption to prevent the addition of cells within the boundary layer, where the mesh was judged to be fine enough already (see *Adapt/Controls...* (p. 2130)).
7. Refine the cells contained in the resulting adaption register (see *Manipulating Adaption Registers* (p. 1343)).
8. Perform successive smoothing and swapping iterations using the *Smooth/Swap Mesh Dialog Box* (p. 2135) (see *Face Swapping* (p. 1353)).

The effect of refining on gradients is evident in the finer mesh ahead of the leading edge of the blade and within the blade passage (*Figure 29.2* (p. 1327)). The finer mesh in the wake region is due to the adaption using isovalues of total pressure.

29.1.2. Adaption Guidelines

The advantages of solution-adaptive refinement, when used properly (as in the turbine cascade example in *Adaption Example* (p. 1326)), are significant. However, this capability must be used carefully to avoid certain pitfalls. Some guidelines for proper usage of solution-adaptive refinement are as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The surface mesh must be fine enough to adequately represent the important features of the geometry.

  For example, it would be bad practice to place too few nodes on the surface of a highly-curved airfoil, and then use solution refinement to add nodes on the surface. The surface will always contain the facets contained in the initial mesh, regardless of the additional nodes introduced by refinement.
- The initial mesh should contain sufficient cells to capture the essential features of the flow field.

  Consider the following example, in which you want to predict the shock forming around a bluff body in supersonic flow. To obtain a reasonable first solution, the initial mesh should contain enough cells and also have sufficient resolution to represent the shape of the body. Subsequent gradient adaption can be used to sharpen the shock and to establish a mesh-independent solution.
- Polyhedral cells are not eligible for adaption. The presence of polyhedral cells in a mesh may or may not limit the eligibility of other cells for adaption, depending on the manner in which the polyhedral cells were created:
  - If the domain was converted to polyhedra (see *Converting the Domain to a Polyhedra* (p. 178)), then no part of the mesh can be adapted (even if hexahedral cells are present in the mesh after conversion).
  - If the polyhedra are a result of converting skewed tetrahedral cells (see *Converting Skewed Cells to Polyhedra* (p. 182)) or converting the transitional cells of a hexcore mesh, then the nonpolyhedral cells may be adapted. The polyhedral cells, however, will be automatically unmarked from the register when adaption is initiated and will remain unchanged.
- Obtain a reasonably well-converged solution before performing an adaption. If you adapt to an incorrect solution, cells will be added in the wrong region of the flow.

  Use careful judgment in deciding how well to converge the solution before adapting, because there is a trade-off between adapting too early to an unconverged solution and wasting time by continuing to iterate when the solution is not changing significantly. This does not directly apply to dynamic adaption, because here the solution is adapted either at every iteration or at every time step, depending on which solver is being used.
- Write a case and data file before starting the adaption process. If you generate an undesirable mesh, you can restart the process with the saved files. This does not directly apply to dynamic adaption, because here the solution is adapted either at every iteration or at every time step, depending on which solver is being used.
- Select suitable variables when performing gradient adaption. For some flows, the choice is clear. For instance, adapting on gradients of pressure is a good criterion for refining in the region of shock waves. In most incompressible flows, however, it makes little sense to refine on pressure gradients. A more suitable parameter in an incompressible flow might be mean velocity gradients. If the flow feature of interest is a turbulent shear flow, it will be important to resolve the gradients of turbulent kinetic energy and turbulent energy dissipation, so these might be appropriate refinement variables. In reacting flows, temperature or concentration (or mole or mass fraction) of reacting species might be appropriate.
- Do not over-refine a particular region of the solution domain. It causes very large gradients in cell volume. Such poor adaption practice can adversely affect the accuracy of the solution.

29.2. Boundary Adaption

This section describes how to perform boundary adaption. For more information, see Boundary Adaption in the Theory Guide.

For additional information, please see the following section:

29.2.1. Performing Boundary Adaption

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.2.1. Performing Boundary Adaption

You can perform the boundary adaption in three different ways based on:

- number of cells

  In this case, the distance of a cell from the boundary is measured in number of cells.
- normal distance

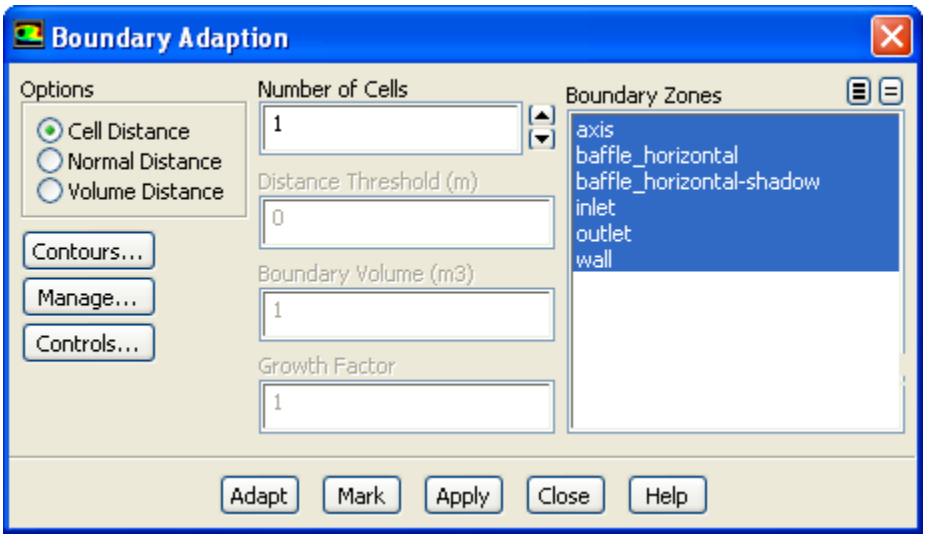
  In this case, the cell refinement is based on the normal distance of a cell from the boundary.
- target boundary volume

  In this case, the cell refinement is based on a target boundary volume and growth factor.

You can use any of these methods in the *Boundary Adaption Dialog Box* (p. 2117) (*Figure 29.3* (p. 1329)).

Adapt → **Boundary...**

Figure 29.3 The Boundary Adaption Dialog Box



29.2.1.1. Boundary Adaption Based on Number of Cells

The procedure for performing adaption based on the distance of a cell from the boundary in terms of the number of cells is as follows:

1. In the *Boundary Adaption Dialog Box* (p. 2117) (*Figure 29.3* (p. 1329)), select **Cell Distance** under **Options**, choose the boundary zones near which you want to refine cells in the **Boundary Zones** list, and click **Apply**.

   This operation is performed to fill the cell distance variable for each cell to be visualized in Step 2.
2. (optional) Click the **Contours...** button to open the *Contours Dialog Box* (p. 1967).
   a. Enable **Filled** contours, disable **Node Values**.
   b. Select **Adaption...** and **Boundary Cell Distance** in the **Contours of** drop-down list.
   c. Select the appropriate surfaces (3D only).
   d. Click **Display** to see the location of cells with each value of boundary cell distance.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By displaying different ranges of values (as described in *Specifying the Range of Magnitudes Displayed* (p. 1390)), you can determine the cell distance of the cells you wish to adapt.

3. Set the **Number of Cells** to the desired value.
   - If you retain the default value of 1, only those cells that have edges (2D) or faces (3D) on the specified boundary zone(s) (i.e., those cells with a boundary cell distance of 1) will be marked or adapted.
   - If you increase the value to 2, cells with a boundary cell distance of 2 will also be marked/adapted, and so on.
4. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click on the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
5. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the refinement immediately.

29.2.1.2. Boundary Adaption Based on Normal Distance

The procedure for performing refinement based on a cell's normal distance from the boundary (i.e., the distance between the centroid of a cell and the boundary) is as follows:

1. In the *Boundary Adaption Dialog Box* (p. 2117), select **Normal Distance** under **Options**, choose the boundary zones near which you want to refine cells in the **Boundary Zones** list, and click **Apply**.

   This operation is performed to fill the cell distance variable for each cell to be visualized in Step 2.
2. (optional) Open the *Contours Dialog Box* (p. 1967) by clicking on the **Contours...** button.
   a. Enable **Filled** contours, disable **Node Values**.
   b. Choose **Adaption...** and **Boundary Normal Distance** in the **Contours of** drop-down list.
   c. Select the appropriate surfaces (3D only).
   d. Click **Display** to see the location of cells with each value of normal distance.

      By displaying different ranges of values (as described in *Specifying the Range of Magnitudes Displayed* (p. 1390)), you can determine the normal distance of the cells you wish to adapt.
3. Set the **Distance Threshold** to the desired value. Cells with a normal distance to the selected boundary zone(s) less than or equal to this value will be marked or adapted.
4. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click on the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
5. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the refinement immediately.

29.2.1.3. Boundary Adaption Based on Target Boundary Volume

This boundary adaption allows you to produce exponentially larger (or smaller) cells as you get further from the boundaries. The cells are marked for refinement based on the following equation:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$V_n > V_{boundary} e^{\alpha d} \quad \textbf{(29–1)}$$

where V_n is the cell volume, $V_{boundary}$ is the specified boundary volume (**Boundary Volume**), α is the exponential growth factor (**Growth Factor**), and d is the normal distance of the cell centroid from the selected boundaries. $V_{boundary} e^{\alpha d}$ is the target volume for a cell.

The procedure for this type of boundary refinement is as follows:

1. In the *Boundary Adaption Dialog Box* (p. 2117), select **Volume Distance** under **Options**, set the **Boundary Volume** and **Growth Factor** to the desired values, select the boundary zones in the **Boundary Zones** list where you want the **Boundary Volume** to be applied, and click **Apply**.

   This operation is performed to fill the cell distance variable for each cell to be visualized in Step 2.

2. (optional) Open the *Contours Dialog Box* (p. 1967) by clicking the **Contours...** button.
   a. Enable **Filled** contours, disable **Node Values**.
   b. Choose **Adaption...** and **Boundary Normal Distance** in the **Contours of** drop-down list.
   c. Select the appropriate surfaces (3D only).
   d. Click **Display** to see the contours of the target volume.

      By displaying different ranges of values (as described in *Specifying the Range of Magnitudes Displayed* (p. 1390)), you can determine the normal distance of the cells you wish to adapt.

   You can modify the values of the inputs (**Boundary Volume**, **Growth Factor**, and/or **Boundary Zones**), click **Apply** in the **Boundary Adaption** dialog box, and then redisplay the contour plot to visualize the modified target volume distribution.

3. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click on the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).

4. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the refinement immediately.

29.3. Gradient Adaption

This section describes how to perform gradient adaption. For more information, see Gradient Adaption in the Theory Guide.

For additional information, please see the following section:
29.3.1. Performing Gradient Adaption

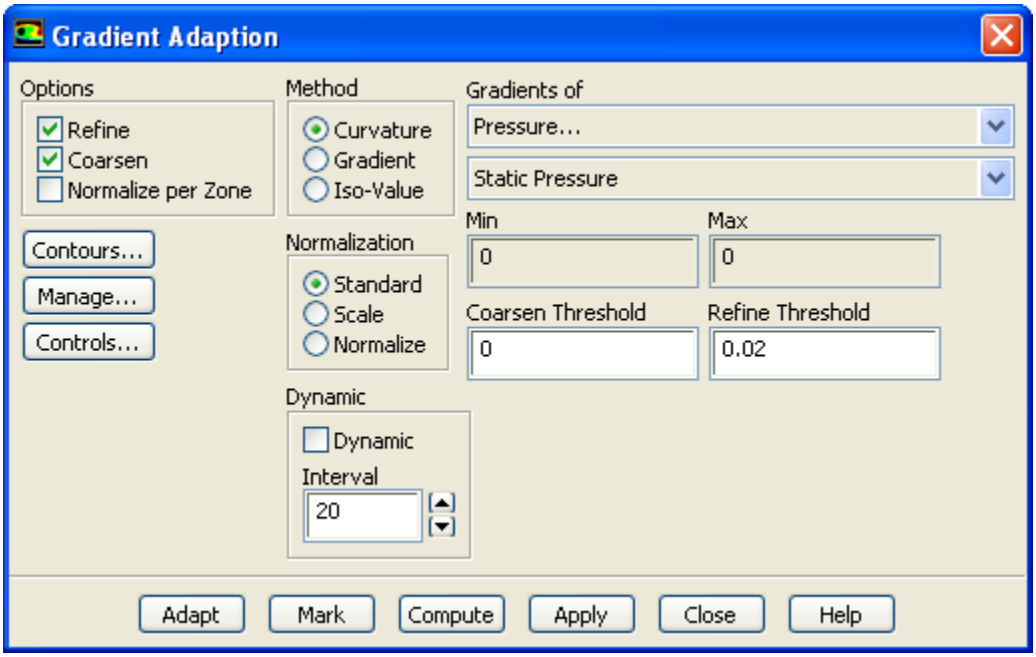
29.3.1. Performing Gradient Adaption

The *Gradient Adaption Dialog Box* (p. 2119) (*Figure 29.4* (p. 1332)) allows you to perform gradient adaption.

Adapt → **Gradient...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.4 The Gradient Adaption Dialog Box



The procedure for performing gradient adaption is as follows:

1. Select the appropriate adaption method.
   - **Curvature** is the default method, and is recommended for problems with smooth solutions.
   - **Gradient** is recommended for problems with strong shocks (e.g., supersonic inviscid flows).
   - **Iso-Value** is recommended for problems where derivatives are not helpful, or when you want to customize the adaption criterion (using custom field functions, user-defined scalars, etc.).
2. Select a **Normalization** method:
   - **Standard**, if normalization of the gradient or curvature is not to be performed.
   - **Scale**, if the gradient or curvature is to be scaled by the average value in the domain.
   - **Normalize**, if the gradient or curvature is to be scaled by the maximum value of the variable in the domain (i.e., the gradient or curvature is bounded by [0, 1]).

   Using either scaling or normalization makes the setting of the refine and coarsen thresholds much simpler, and almost independent of the current solution and specific problem.

   This is especially important when using the automated dynamic adaption process.
3. Select the required solution variable in the **Gradients of** drop-down list.
4. Click **Compute**.
5. Click **Contours...** to open the *Contours Dialog Box* (p. 1967).
   a. Enable **Filled** contours, disable **Node Values**, and select**Adaption...** and **Existing Value** in the **Contours of** drop-down lists.
   b. Select the appropriate surfaces (3D only).
   c. Click **Display** to see the location of cells with each curvature value.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By displaying different ranges of values (as described in *Specifying the Range of Magnitudes Displayed* (p. 1390)), you can determine the range of curvatures for which you want to adapt cells.

If you are using normalization, the range for the curvatures of any variable will always be [0, 1].

6. Set the values for **Refine Threshold**.

   Cells with gradient values above this value will be either marked or refined.

7. Select the **Normalize per Zone** option for cases where different flow conditions exist for different zones.

   This approach of *zonal normalization* normalizes (i.e., scales) each zone of the domain, in contrast to normalization on the whole domain. This approach is useful for dynamic adaption (see *Dynamic Gradient Adaption* (p. 1333) for details), where you want to solve the flow problem involving different flow intensities in the different cell zones.

   If you use gradient adaption for the whole domain, the small gradients may be neglected in comparison to large gradients depending on the adaption threshold. Activating **Normalize per Zone** in the *Gradient Adaption Dialog Box* (p. 2119) will scale or normalize each zone independently, which means the strongest gradient for each zone is considered separately for adaption of that zone.

   Note

   If you expect gradients of different intensities throughout the domain and you want to resolve them, separate the domain into different zones for precise zonal normalization. This approach is referred as zonal adaption.

8. If you want to coarsen the mesh, set the **Coarsen Threshold** to a non-zero value. Cells with gradient values below the specified value will be either marked or coarsened.
9. To set adaption options (described in *Adapt/Controls...* (p. 2130)), click **Controls...** to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
10. To mark the cells for adaption (refinement/coarsening), click **Mark**. You can then place the cells in an adaption register, which can be manipulated (as described in *Manipulating Adaption Registers* (p. 1343)). To perform the adaption immediately, click **Adapt**.

Note

To disable refinement, coarsening, or marking for refinement/coarsening, disable the **Refine** or **Coarsen** option before marking or adapting.

29.4. Dynamic Gradient Adaption

This section describes how to perform dynamic gradient adaption. For more information, see Dynamic Gradient Adaption in the Theory Guide.

For additional information, please see the following section:

29.4.1. Dynamic Gradient Adaption Approach

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.4.1. Dynamic Gradient Adaption Approach

The dynamic gradient adaption executes the gradient adaption automatically. Though all options of gradient adaption are valid for the dynamic gradient adaption, some specific settings are recommended:

In the **Gradient Adaption** dialog box:

- Enable the **Refine** and **Coarsen** options.
- The **Normalize per Zone** enables zonal normalization for the dynamic adaption. See *Performing Gradient Adaption* (p. 1331) (step *7* (p. 1333)) for details.
- For **Normalization**, use either the **Scale** or the **Normalize** option.

  *The non-normalized values of the gradient or the curvature of a variable (obtained by selecting **Standard** for the **Normalization**) are generally strongly solution dependent, and therefore would require re-adjustment of the **Coarsen Threshold** and **Refine Threshold** as the solution proceeds.*
- For dynamic adaption, scaling is preferred if you wish to resolve regions of small values of the gradient (or curvature/isovalues) accurately, in addition to the region of highest gradient (or curvature/isovalues).

  Scaling does not take very high values of the gradient or curvature into account to the degree that normalization does.
- The starting values for **Refine Threshold** and **Coarsen Threshold** are 1e10 and 0 respectively.

  The more refinement you want, the smaller these values should be.
- Specify the **Interval** between two consecutive automatic mesh adaptions. Depending on whether you are performing a steady-state or a time-dependent solution, specify **Interval** in iterations or time steps, respectively.

  This value depends on the type of problem solved and the time step used (where applicable). For steady-state problems, values of 100 or higher are reasonable. For time-dependent problems, values of 10 or lower are often required.

  If you are using the density-based explicit solver with explicit transient formulation, your input will be in number of iterations.

In the **Mesh Adaption Controls** dialog box:

- Set values for **Min # of Cells**, **Max # of Cells**, **Max Level of Refine**, or **Min Cell Volume**.

  The limits for the **Min # of Cells** and **Max # of Cells** can affect the **Coarsen Threshold** and **Refine Threshold** values. If either the **Min # of Cells** or the **Max # of Cells** are violated, the **Coarsen Threshold** or the **Refine Threshold** are adjusted to fulfill the limits for the **Min # of Cells** or the **Max # of Cells**.
- The default value for **Max Level of Refine** is 2, which is a good start for most problems. If required, you can increase this value.

Important

Even in a 2D problem, the default value of 2 can increase the number of cells by a factor of 16 in the adapted regions. A value of zero leaves this parameter unbounded: in this case, you should use a suitable limit for **Min Cell Volume**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.4.1.1. Examples of Dynamic Gradient Adaption

Example 1: Steady-state problem

Consider a supersonic flow over a blunt body. To determine the wave drag for such problem, first resolve the shock wave. Start with a coarse mesh and set up dynamic adaption. As you start iterating the solution, the solver will produce a blurred shock, probably in an incorrect location. After the adaptions, the shock will become sharper and move into the correct location.

Example 2: Time-dependent problem

Consider a traveling shock wave. To determine the precise pressure amplitudes and arrival times at a number of locations, you need to resolve the shock wave over the time, so that you can maintain the correct shock strength and its location. Dynamic adaption is efficient in this case, as it refines the mesh near the shock, and at the same time, it coarsens the mesh wherever needed.

29.5. Isovalue Adaption

This section describes how to perform isovalue adaption. For more information, see Isovalue Adaption in the Theory Guide.

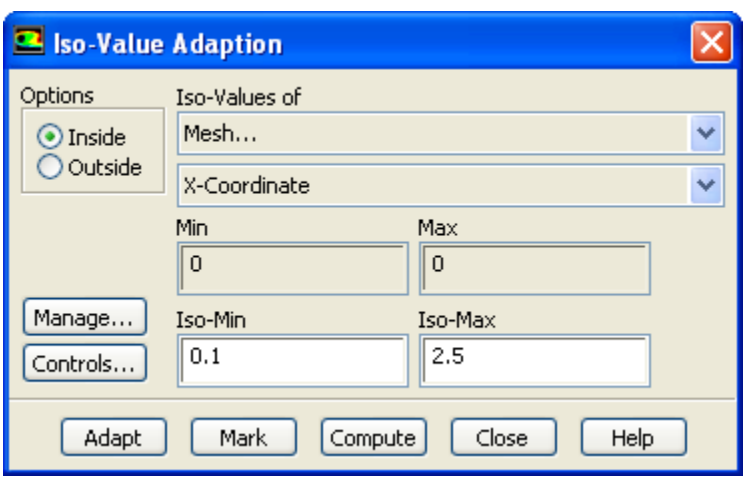
For additional information, please see the following section:
29.5.1. Performing Isovalue Adaption

29.5.1. Performing Isovalue Adaption

You can perform isovalue adaption in the *Iso-Value Adaption Dialog Box* (p. 2121) (*Figure 29.5* (p. 1335)).

Adapt → **Iso-Value...**

Figure 29.5 The Iso-Value Adaption Dialog Box



The general procedure for performing isovalue adaption is as follows:

1. Select the desired solution variable in the **Iso-Values of** drop-down lists and click **Compute** to update the **Min** and **Max** fields.
2. Choose the **Inside** or **Outside** option and set the **Iso-Min** and **Iso-Max** values.
   - If you choose **Inside**, cells with isovalues between **Iso-Min** and **Iso-Max** will be marked or refined.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you choose **Outside**, cells with isovalues less than **Iso-Min** or greater than **Iso-Max** will be marked or refined.

3. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
4. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the refinement immediately.

29.6. Region Adaption

This section describes how to perform region adaption. For more information, see Region Adaption in the Theory Guide.

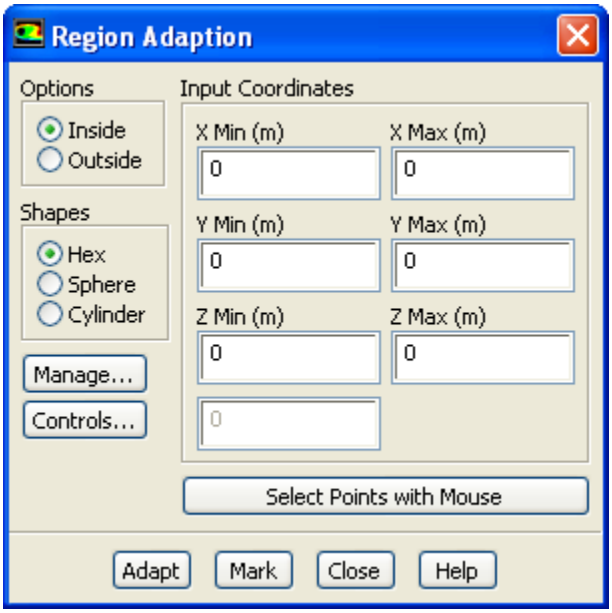
For additional information, please see the following section:
29.6.1. Performing Region Adaption

29.6.1. Performing Region Adaption

You will perform region adaption in the *Region Adaption Dialog Box* (p. 2122) (*Figure 29.6* (p. 1336)).

Adapt → **Region...**

Figure 29.6 The Region Adaption Dialog Box



The procedure for performing isovalue adaption is as follows:

1. In the *Region Adaption Dialog Box* (p. 2122), select the **Inside** or **Outside** option.
   - If you choose **Inside**, cells with centroids within the specified region will be marked or refined.
   - If you choose **Outside**, cells with centroids outside the specified region will be marked or refined.
2. Specify the shape of the region.

   In 2D, you may specify a **Quad** (i.e., a quadrilateral), **Circle**, or **Cylinder** by making a selection from the **Shape** group box. In 3D, your options include **Hex** (i.e., a hexahedron), **Sphere**, or **Cylinder**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the region by entering values into the dialog box or by using the mouse.

   In the dialog box, the inputs are as follows:

   - To define a hexahedron or quadrilateral, enter the coordinates of two points defining the diagonal of the box.

     For a hexahedron, define **X Min**, **Y Min**, and **Z Min**, as well as **X Max**, **Y Max**, and **Z Max**. For a quadrilateral, define **X Min** and **Y Min**, as well as **X Max** and **Y Max**.
   - To define a sphere or circle, enter the values for the **Radius** and the coordinates of its center: **X Center**, **Y Center**, and **Z Center** for a sphere, or **X Center** and **Y Center**for a circle.
   - To define a cylinder, enter the value for the **Radius** and the minimum and maximum coordinates defining the cylinder axis: **X-Axis Min**, **Y-Axis Min**, and **Z-Axis Min**, as well as **X-Axis Max**, **Y-Axis Max**, and **Z-Axis Max** for 3D, or **X-Axis Min** and **Y-Axis Min**, as well as **X-Axis Max** and **Y-Axis Max** for 2D. In 2D, this will be the width of the resulting rectangle.
4. To define the region using the mouse, click on the **Select Points with Mouse** button. Using the right mouse button, select the input coordinates from a display of the mesh or solution field. After selecting the points, the values will be loaded automatically into the appropriate fields in the dialog box. See *Controlling the Mouse Button Functions* (p. 1427) for details about mouse button functions.

   You have the option of editing these values before marking or adapting.

   - To define a hexahedron or quadrilateral, select the two points of the diagonal in any order.
   - To define a sphere or circle, first select the location of the centroid and then select a point that lies on the sphere/circle (i.e., a point that is one radius away from the centroid).
   - To define a cylinder, first select the two points that define the cylinder axis and then select a point that is one radius away from the axis.
5. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
6. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the refinement immediately.

29.7. Volume Adaption

This section describes how to perform volume adaption. For more information, see Volume Adaption in the Theory Guide.

For additional information, please see the following section:
29.7.1. Performing Volume Adaption

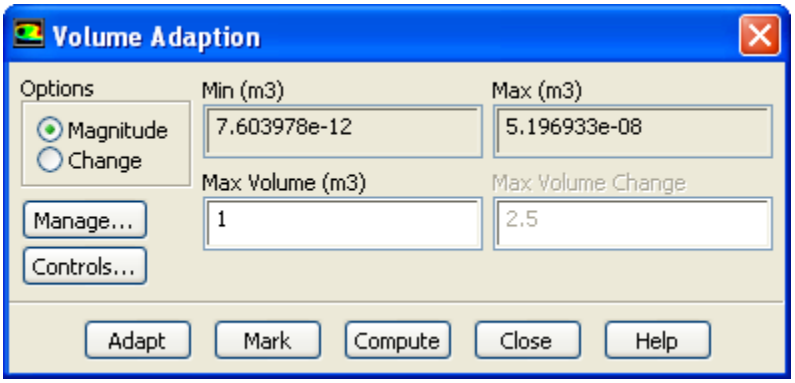
29.7.1. Performing Volume Adaption

You will perform volume adaption in the *Volume Adaption Dialog Box* (p. 2124) (*Figure 29.7* (p. 1338)).

Adapt → **Volume...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.7 The Volume Adaption Dialog Box



The procedure for performing volume adaption is as follows:

1. In the *Volume Adaption Dialog Box* (p. 2124), specify whether you want to adapt based on volume magnitude or volume change by selecting the **Magnitude** or **Change** option.
2. Click **Compute** to update the **Min** and **Max** fields. These fields will show the range of cell volumes or cell volume changes (defined in Volume Adaption Approach in the Theory Guide), depending on your selection in step 1.
3. Set the **Max Volume** or **Max Volume Change** value.
   a. If you have chosen to adapt based on volume **Magnitude**, cells that have volumes greater than **Max Volume** will be marked or refined.
   b. If you are adapting based on volume **Change**, cells with volume changes greater than **Max Volume Change** will be marked or refined.
4. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
5. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated, as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the refinement immediately.

29.8. Yplus/Ystar Adaption

This section describes how to perform Yplus/Ystar adaption. For more information, see Yplus/Ystar Adaption in the Theory Guide.

For additional information, please see the following section:
29.8.1. Performing Yplus or Ystar Adaption

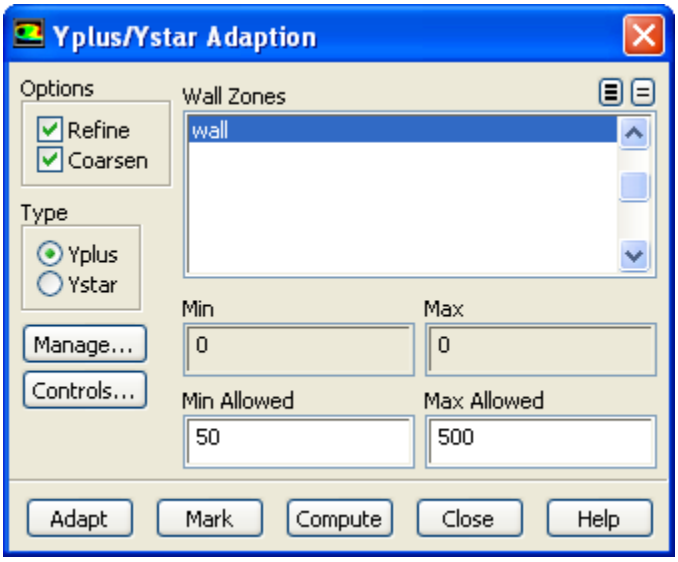
29.8.1. Performing Yplus or Ystar Adaption

You will perform Yplus or Ystar adaption in the *Yplus/Ystar Adaption Dialog Box* (p. 2126) (*Figure 29.8* (p. 1339)).

Adapt → **Yplus/Ystar...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.8 The Yplus/Ystar Adaption Dialog Box



The procedure for performing y^+ or y^* adaption is as follows:

1. In the *Yplus/Ystar Adaption Dialog Box* (p. 2126), select **Yplus** or **Ystar** as the adaption **Type**.
   - Select **Yplus** if you are using the enhanced wall treatment.
   - If you are using wall functions, you can select either type.
2. Select the wall zones for which you want boundary cells to be marked or adapted in the **Wall Zones** list, and click **Compute** to update the **Min** and **Max** fields. The values displayed are the minimum and maximum values for all wall zones, not just of those selected.
3. Set the **Min Allowed** and **Max Allowed**. Cells with y^+ or y^* values below **Min Allowed** will be coarsened or marked for coarsening, and cells with y^+ or y^* values above **Max Allowed** will be refined or marked for refinement.
4. (optional) If you want to set any adaption options (described in *Adapt/Controls...* (p. 2130)), click the **Controls...** button to open the *Mesh Adaption Controls Dialog Box* (p. 2130).
5. Click **Mark** to mark the cells for adaption (refinement/coarsening) by placing them in an adaption register (which can be manipulated as described in *Manipulating Adaption Registers* (p. 1343)), or click **Adapt** to perform the adaption immediately.

To disable refinement or coarsening, or marking for refinement or coarsening, turn off the **Refine** or **Coarsen** option before marking or adapting.

29.9. Anisotropic Adaption

This section describes how to perform anisotropic adaption. For more information, see Anisotropic Adaption in the Theory Guide.

For additional information, please see the following sections:

29.9.1. Limitations of Anisotropic Adaption
29.9.2. Performing Anisotropic Adaption

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.9.1. Limitations of Anisotropic Adaption

Since anisotropic adaption is available only for specific cell types, the following limitations exist:

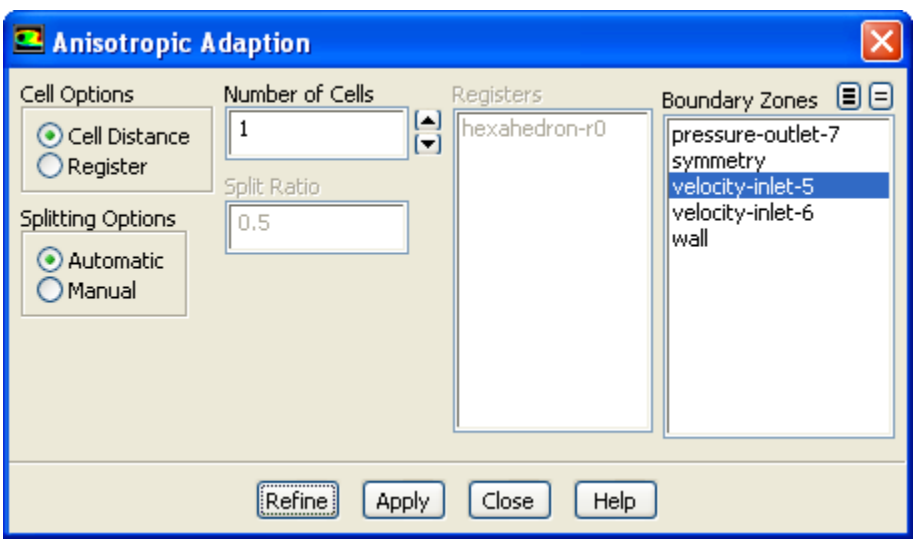
- It is only available in 3D.
- It only works for hexahedral cells or prism cells.
- Each cell can only be split into two, with a given splitting ratio in the normal direction of the boundary face. Multiple layers can be achieved by multiple refinement.
- Each cell to be split can only be reached once from any of the boundary faces; otherwise, the refinement will not be processed.
- Unlike other adaption functionalities, the subdivided cells cannot be coarsened again, because all the cells that are adjacent to the refined cells are converted into polyhedral cells.

29.9.2. Performing Anisotropic Adaption

You will perform anisotropic adaption in the *Anisotropic Adaption Dialog Box* (p. 2127) (*Figure 29.9* (p. 1340)).

Adapt → **Anisotropic...**

Figure 29.9 The Anisotropic Adaption Dialog Box



The procedure for performing anisotropic adaption is as follows:

1. In the *Anisotropic Adaption Dialog Box* (p. 2127), select **Cell Distance** or **Register** from the **Cell Options** group box. This allows you to control the marking of boundary layer cells. Select **Cell Distance** and enter the **Number of Cells** to be adapted and marked using the distance from the boundary zone. Select **Register** to adapt and mark cells using an existing **Register**.
2. The **Splitting Options** control how the splitting ratio is computed. It is defined as follows:

   $$\text{splitting ratio} = \frac{\text{height of the splitting point to the base face}}{\text{original height of the cell}}$$

   By default, **Automatic** is selected, resulting in the ratio being computed automatically from the mesh. If you selected **Manual**, enter the desired **Split Ratio**. If you choose to compute the split ratio auto-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

matically, the split ratio of the first layer is computed, and it may be 0.5 if the original cells are uniformly distributed, resulting in the height of the first layer being the same as the height of the second layer.

Important

Note that the **Split Ratio** that you enter is only applicable to the first layer, and all the other layers are split with a ratio of 0.5.

3. For the **Cell Distance** option, one or more boundary face zones must be selected before marking the cells for refinement.
4. For the **Register** option, select a register in the list. Note that one or more boundary face zones must be selected before doing the refinement.
5. Click **Refine** to refine the marked cells.

29.10. Geometry-Based Adaption

This section describes how to perform geometry-based adaption. For more information, see Geometry-Based Adaption in the Theory Guide.

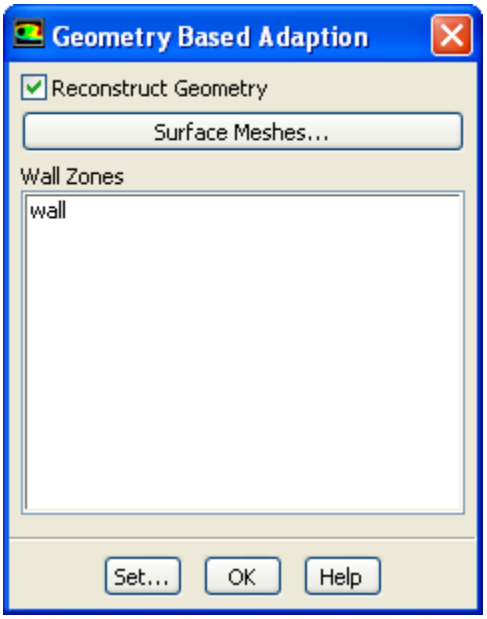
For additional information, please see the following section:
29.10.1. Performing Geometry-Based Adaption

29.10.1. Performing Geometry-Based Adaption

The *Geometry Based Adaption Dialog Box* (p. 2132) (*Figure 29.10* (p. 1341)) allows you to reconstruct the geometry while performing boundary adaption.

Adapt → **Geometry...**

Figure 29.10 The Geometry Based Adaption Dialog Box

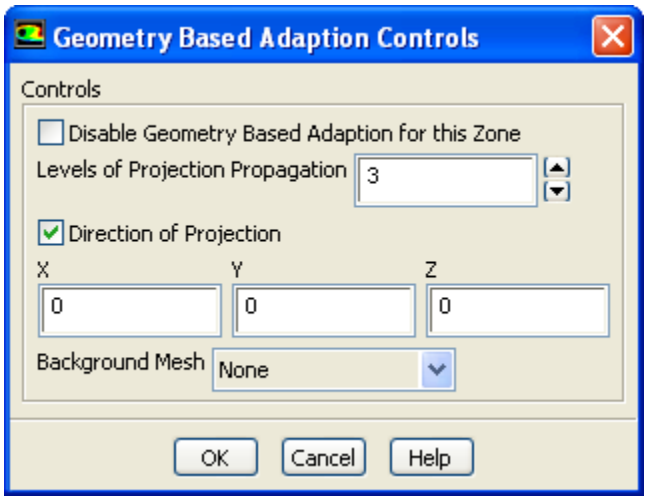


The procedure for performing geometry-based adaption is as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable the **Reconstruct Geometry** option.
2. Under **Wall Zones**, select the zone you want to adapt and click **Set...**. The *Geometry Based Adaption Controls Dialog Box* (p. 2133) will open.

Figure 29.11 The Geometry Based Adaption Controls Dialog Box



In the *Geometry Based Adaption Controls Dialog Box* (p. 2133), set the following parameters:

- Specify **Levels of Projection Propagation** to indicate the number of layers of the nodes you want to project.
- Enable **Direction of Projection** and specify the directions in which you want to project the nodes.

  This will activate the parameters X, Y, and Z. If you want node projection in the X direction, specify X=1. If you do not activate this option, the node projection will take place at the nearest point.
- (optional) If you have a fine surface mesh for the geometry, you can use the **Background Mesh** option to load the surface mesh as a background mesh. This will project the nodes based on the background mesh and reconstruct the geometry more accurately.
- To disable the geometry reconstruction for any zone in the domain, activate **Disable Geometry Based Adaption for this Zone**.

3. To disable geometry-based adaption for the whole domain, disable **Reconstruct Geometry**.

After setting the parameters for geometry-based adaption, proceed to perform mesh adaption.

29.11. Registers

This section describes how to use registers for adaption. For more information, see Registers in the Theory Guide.

For additional information, please see the following sections:

29.11.1. Manipulating Adaption Registers
29.11.2. Modifying Adaption Marks
29.11.3. Displaying Registers
29.11.4. Adapting to Registers

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.11.1. Manipulating Adaption Registers

You can manipulate, delete, and display adaption registers by marking cells for adaption. Since these registers are used to adapt the mesh, the ability to manipulate them provides additional control over the adaption process.

Management of adaption registers is performed in the *Manage Adaption Registers Dialog Box* (p. 2129) (*Figure 29.12* (p. 1343)). You can also open this dialog box by clicking on the **Manage...** button in any of the adaption dialog boxes.

Adapt → **Manage...**

Figure 29.12 The Manage Adaption Registers Dialog Box

You can modify and manipulate adaption registers by:

- changing the register types
- combining the registers
- deleting the registers

29.11.1.1. Changing Register Types

If the adaption register is converted to a mask, the cells marked for refinement are ACTIVE, and all other cells are INACTIVE (i.e., the cells marked for coarsening are ignored). Generally, the adaption registers converted to masks are those that are generated by adaption functions that mark cells exclusively for refinement, such as region or isovalue adaption functions. The other major difference between adaption and mask registers is the manner in which they are combined.

To change the type of one or more registers from adaption to mask, or vice versa, do the following:

1. Choose the register(s) in the **Registers** list.
2. Click the **Change Type** button under **Register Actions**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new type of the register (if multiple registers are selected, the most recently selected or deselected register) will be shown as the **Type** under **Register Info**. Select each register individually to see what its current type is.

29.11.1.2. Combining Registers

After the individual adaption registers have been created and appropriately modified, they are combined to create hybrid adaption functions.

1. Any number of registers can be combined in the following manner:
   - All adaption registers are combined into a new adaption register.
   - All mask registers are combined into a new mask register.
   - The new adaption and mask registers are combined.
2. Any number of adaption registers can be combined in the following manner:
   - If the cell is marked for refinement in any of the registers, mark the cell for refinement in the new register (bitwise OR).
   - If the cell is marked for coarsening in all of the registers, mark the cell for coarsening in the new register (bitwise AND).
3. The mask registers are combined in a manner similar to the refinement marks. If any cell is marked ACTIVE, the cell in the new register is marked ACTIVE (bitwise OR).
4. Finally, in the combination of an adaption and mask register, only cells that are marked in the mask register can have an adaption mark in the combined register (bitwise AND).

For example, creating an adaption function based on pressure gradient may generate cells marked for refinement and coarsening throughout the entire solution domain. If this register is then combined with a mask register created from cells marked inside a sphere, only the cells inside the sphere will be marked for refinement or coarsening in the new register.

Note

The effect of masks depends on the order in which they are applied.

For example, consider two adjacent, circular masks. Applying one mask to the adaption register and then applying the other mask to the result of the first combination would give a much different result than applying the combination of the two masks to the initial adaption register. The second combination results in a greater possible number of marked cells.

To combine two or more registers, do the following:

1. Choose the registers in the **Registers** list.
2. Click the **Combine** button under **Register Actions**.

The selected registers will remain intact, and the register(s) resulting from the combination will be added to the **Registers** list. In some instances, three new registers may be created:

- a combination of the adaption registers
- a combination of the mask registers
- a combination of the two combined registers

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about combining registers, see Adaption Registers in the Theory Guide.

29.11.1.3. Deleting Registers

The primary reason for deleting registers is to discard unwanted adaption registers. This will reduce confusion and the possibility of generating undesired results by selecting these discarded registers. In addition, only 32 adaption registers can exist at one time. Therefore, you should discard unwanted registers to make room for new ones. You can delete any number of adaption registers.

To permanently remove one or more registers, do the following:

1. Choose the register(s) in the **Registers** list.
2. Click the **Delete** button under **Register Actions**.

29.11.2. Modifying Adaption Marks

The adaption marks are the identifiers that designate whether a cell should be refined, coarsened, or neutral. The operations used for modifying the adaption marks are:

- **Exchange**: This changes the cells marked for refinement into cells marked for coarsening, and all cells originally marked for coarsening into cells marked for refinement. This operation is applied to adaption registers that have only refinement marks.

  For example, the exchange operation can be used to coarsen a rectangular region. First, create an adaption register that marks a rectangular region of cells for refinement. Then use the **Exchange** operation to modify the cell marks, creating a rectangular region with cells marked for coarsening.
- **Invert**: This operation can only be used with mask registers. It toggles the mask markings, i.e., all cells marked ACTIVE are switched to INACTIVE, and all cells marked INACTIVE are switched to ACTIVE.

  For example, if you generate a mask that defines a circular region, you can quickly modify the mask to define the region outside of the circle using the **Invert** operation.
- **Limit**: This operation applies the present adaption volume limit to the selected adaption register. For information on adaption limits, see *Mesh Adaption Controls* (p. 1347). You generally use this operation to determine the effect of the present limits on the adaption process. You can use the volume limit to create a uniform mesh by setting the limit to refine only the large cells. After all the cells have reached a uniform size, you can continue the refinement process to the desired resolution.
- **Fill**: This operation marks the cells in the adaption register that are not marked for refinement. You can use the **Fill** operation to combine multiple registers to make a new register.

  Note the following:

  – When you combine registers, a cell will be marked for coarsening only if it is marked for coarsening in all of the registers.

  – If you create an adaption register with an operation that only marks cells for refinement, but you do not want to prohibit coarsening, use the **Fill** operation before combining the register with any other registers.

The process for modifying adaption marks is as follows:

1. Choose the register(s) in the **Registers** list.
2. Click the **Exchange**, **Invert**, **Limit**, or **Fill** button under **Mark Actions**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.11.3. Displaying Registers

Viewing the cell markings is often helpful in the process of creating hybrid adaption functions. You can plot a marker at the cell centroid and/or a wireframe of the cell to view the state of the cell. By default, the cells marked for refinement are colored in red, and the cells marked for coarsening are marked in cyan. In addition, cells marked ACTIVE in a mask register are also colored red. These are the cells that are marked for adaption, but the final number of cells added or subtracted from the mesh depends on the adaption limits and the mesh characteristics.

To display a register, do the following:

1. Choose the register in the **Registers** list.
2. Set the display options by clicking on the **Options...** button.
3. Click the **Display** button.

29.11.3.1. Adaption Display Options

Various aspects of the adaption register display can be modified, such as the wireframe visibility and shading, marker visibility, color, size, and symbol. Also, you can select either surface or zone meshes for the display.

The adaption register display capability allows you to view the cells that are flagged for adaption.

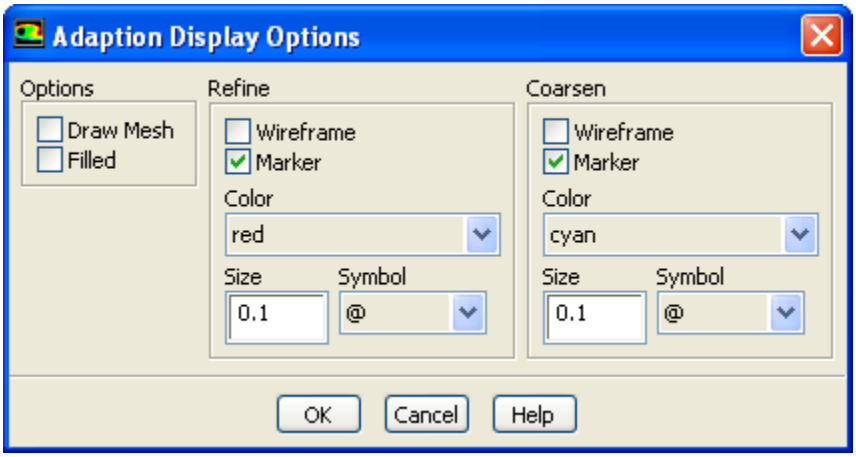
- Depending on the dimension of the problem and the number of flagged cells, you can customize the adaption display options. The most common method for viewing flagged cells in 2D is to draw the mesh and filled wireframes, but this is impractical in 3D. In three dimensions, you can plot the centroid markers of the cells with the mesh of selected boundary zones.
- You can use markers and/or wireframes to display the flagged cells in an adaption or mask register. The marker is a symbol placed at the centroid of the cell. There is a refine marker and a coarsen marker. You can change the symbol, color, and size of these markers. A wireframe is composed of the edges of the triangle or tetrahedron. Its color is the same as the respective marker color, and can be filled, if required.
- Portions of the mesh can be drawn with the marker symbols or wireframes to aid in evaluating the location of marked cells.

All of these options are set in the *Adaption Display Options Dialog Box* (p. 2134) (*Figure 29.13* (p. 1347)). You can also open this dialog box by clicking on the **Options...** button in the *Manage Adaption Registers Dialog Box* (p. 2129).

Adapt → **Display Options...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.13 The Adaption Display Options Dialog Box



- To enable or disable the display of wireframes for cells marked for refinement/coarsening, turn the **Wireframe** option on or off under **Refine** and/or **Coarsen**. To draw filled wireframes (i.e., using a solid color, instead of the outline) turn on the **Filled** option.
- To enable or disable the display of markers for cells marked for refinement/coarsening, turn the **Marker** option on or off under **Refine** and/or **Coarsen**. Use markers to specify their size in the **Size** field, and their symbol in the **Symbol** drop-down list.
- To change the color of the refine or coarsen markers/wireframes, select a new color in the **Color** drop-down list under **Refine** or **Coarsen**. By default, refine markers/wireframes are red and coarsen markers/wireframes are cyan.
- To include portions of the mesh in the register display, enable the **Draw Mesh** option. The *Mesh Display Dialog Box* (p. 1635) will open automatically, where you can set the mesh display parameters. When you click **Display** in the *Manage Adaption Registers Dialog Box* (p. 2129), the mesh display, as defined in the *Mesh Display Dialog Box* (p. 1635), will be included in the register display.

29.11.4. Adapting to Registers

These register tools provide you with the ability to create hybrid adaption functions customized to your flow-field application. The customized adaption function is used to direct the refinement and coarsening of the mesh.

To perform the adaption, follow these steps:

1. Choose the register in the **Registers** list.
2. Click the **Adapt** button.

29.12. Mesh Adaption Controls

ANSYS FLUENT allows you to:

- Place restrictions on the cell zones.
- Limit adaption by cell volume or volume weight.
- Limit the total number of cells that can be produced from the adaption process.
- Modify the intensity of the volume weighting in the gradient function.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

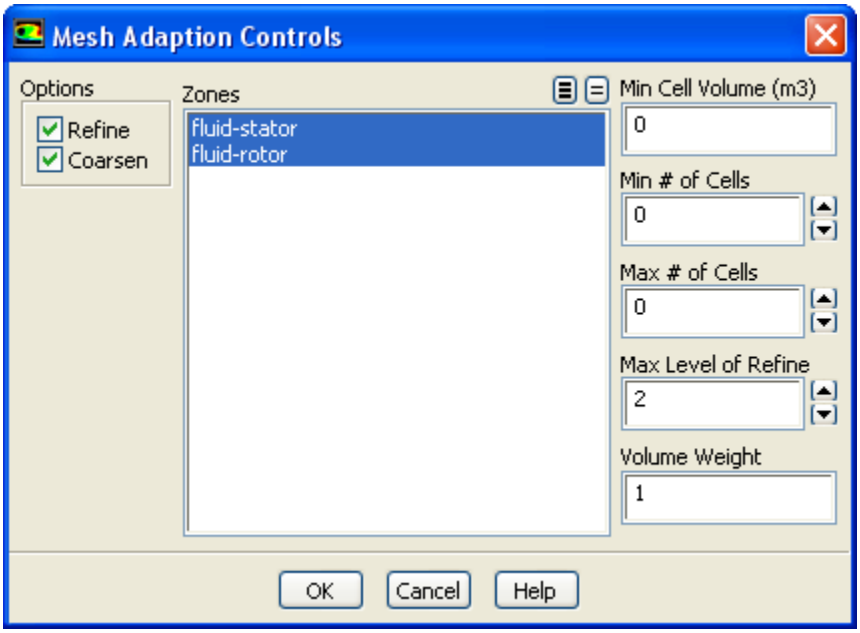
- Restrict the adaption process to refinement and/or coarsening, and control which nodes are eligible for possible elimination from the mesh during coarsening.

The parameters controlling the aspects of adaption are set in the *Mesh Adaption Controls Dialog Box* (p. 2130) (*Figure 29.14* (p. 1348)).

You can open this dialog box by using the **Adapt/Controls...** menu item or by clicking the **Controls...** button in any of the adaption dialog boxes.

Adapt → **Controls...**

Figure 29.14 The Mesh Adaption Controls Dialog Box



Note

Write a case and data file before starting the adaption process. Then, if you generate an undesirable mesh, you can restart the process with the saved files.

For additional information, please see the following sections:

29.12.1. Limiting Adaption by Zone
29.12.2. Limiting Adaption by Cell Volume or Volume Weight
29.12.3. Limiting the Total Number of Cells
29.12.4. Controlling the Levels of Refinement During Hanging Node Adaption

29.12.1. Limiting Adaption by Zone

You can limit the adaption process to specified cell zones. The cells composing the fluid and solid regions of the analysis generally have very different resolution requirements and error indicators. Limiting the adaption to a specific cell zone and use different adaption functions to create the optimal mesh.

To limit the adaption to a particular cell zone (or to particular cell zones), select the cell zones in which you want to perform adaption in the **Zones** list. By default, adaption will be performed in all cell zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.12.2. Limiting Adaption by Cell Volume or Volume Weight

The minimum cell volume limit restricts the refinement process to cells with volumes greater than the limit. Use this to initiate the refinement process on larger cells, gradually reducing the limit to create a uniform cell size distribution. Set this limit in the **Min Cell Volume** field. The input that you will give in this field for a 2D axisymmetric problem will be interpreted as the minimum cell area.

In addition, the gradient volume weight can be modified. A value of zero eliminates volume weighting, a value of unity uses the entire volume, and values between 0 and 1 scale the volume weighting. Set this value in the **Volume Weight** field. For more information, see Gradient Adaption Approach in the Theory Guide.

29.12.3. Limiting the Total Number of Cells

The maximum number of cells is a restriction that prevents ANSYS FLUENT from creating more cells than required for the present analysis. In addition, it saves the time you would spend waiting for the mesh adaption process to complete the creation of these cells. However, this premature termination of the refinement process can produce undesirable mesh quality depending on the order in which the cells were visited, which is based on the cell arrangement in memory (random).

During the dynamic gradient adaption, the resulting number of cells after adaption is estimated. If this number exceeds the maximum number of cells, both the **Coarsen Threshold** and the **Refine Threshold** are updated. This is done to ensure the best possible mesh resolution with the specified number of cells. You can also specify the minimum number of cells. This is helpful if strong structures of the flow that were resolved with the adaption vanished (e.g., left the domain) and you want to resolve the remaining weaker ones. This would otherwise require modifying the **Coarsen Threshold** and the **Refine Threshold**.

You can set the total number of cells allowed in the mesh in the **Max # of Cells** field. The minimum number of cells in the mesh can be set in the **Min # of Cells** field. The default values of zero places no limits on the number of cells.

29.12.4. Controlling the Levels of Refinement During Hanging Node Adaption

You can control the number of levels of refinement used to split cells during nonconformal adaption by setting the **Max Level of Refine**. The default value of 2 is a good start for most problems. If this is not sufficient, you can increase this value.

Note

Even in a 2D problem, the default value of 2 can increase the number of cells by a factor of 16 in the adapted regions.

A value of zero leaves this parameter unbounded, and you should use a suitable limit for **Min Cell Volume**. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide. For guidelines for limiting cell sizes and number of cells during dynamic gradient adaption, see *Dynamic Gradient Adaption Approach* (p. 1334).

29.13. Improving the Mesh by Smoothing and Swapping

Smoothing and face swapping are tools that complement mesh adaption by increasing the quality of the final numerical mesh. Smoothing repositions the nodes, and face swapping modifies the cell connectivity to achieve these improvements in quality.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Face swapping is applicable only to meshes with triangular or tetrahedral cells.

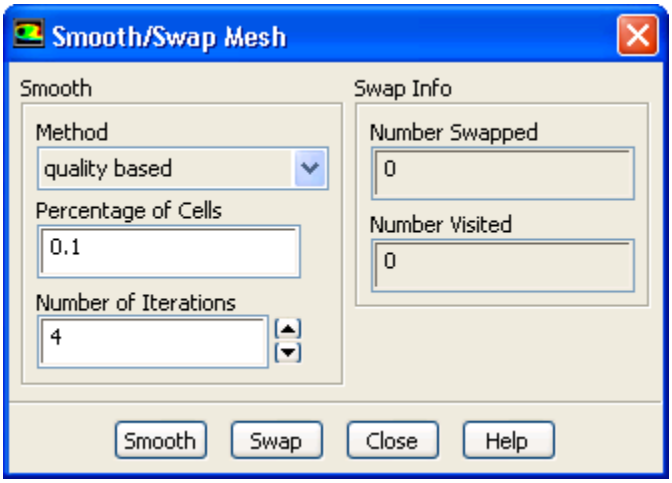
Important

Face swapping and most methods of smoothing are available only for serial cases; only quality-based smoothing can be used for parallel cases.

Both smoothing and swapping are performed using the *Smooth/Swap Mesh Dialog Box* (p. 2135) (*Figure 29.15* (p. 1350)).

Adapt → Smooth/Swap...

Figure 29.15 The Smooth/Swap Mesh Dialog Box



For additional information, please see the following sections:

29.13.1. Smoothing
29.13.2. Face Swapping
29.13.3. Combining Skewness-Based Smoothing and Face Swapping

29.13.1. Smoothing

The three smoothing methods that are available in ANSYS FLUENT are:

- quality-based smoothing

  This method is recommended for all types of meshes.

- Laplacian smoothing

  This method can be applied to all types of meshes, but it is recommended to use it for quadrilateral and hexahedral meshes.

- skewness-based smoothing

  This method is recommended for triangular and tetrahedral meshes, and can be used alternatively with face swapping (see *Combining Skewness-Based Smoothing and Face Swapping* (p. 1354)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.13.1.1. Quality-Based Smoothing

When you use the quality-based smoothing method, ANSYS FLUENT will divide the mesh into a number of "bins", each of which contain a certain number of cells. Improvements are attempted on the cells in those bins that exhibit the lowest orthogonal quality (as defined in *Mesh Quality* (p. 144)). As part of this method, you specify the percentage of the total number of cells, in order to determine how many bins are modified.

Note that the method employed during quality-based smoothing is similar to when you use the `mesh/repair-improve/improve` text command. The advantage of using the **Smooth/Swap Mesh** dialog box rather than the text command is that you can control the percentage of the cells that ANSYS FLUENT attempts to improve.

To perform quality-based smoothing, do the following steps:

1. In the *Smooth/Swap Mesh Dialog Box* (p. 2135) (*Figure 29.15* (p. 1350)), select **quality based** in the **Method** drop-down list in the **Smooth** group box.
2. Enter the **Percentage of Cells** to which you want improvements made.

   Important

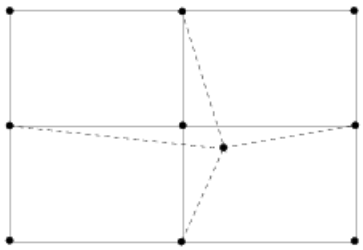
   Quality-based smoothing will be CPU intensive if you specify a large value for the **Percentage of Cells**. It is recommended that you enter a small value initially, and then perform the smoothing process multiple times, if necessary. Note that the maximum percentage allowed is 10%.

3. Specify the number of successive smoothing sweeps to be performed on the mesh in the **Number of Iterations** number-entry box. The default value is 4.
4. Click the **Smooth** button.

29.13.1.2. Laplacian Smoothing

When you use this method, a Laplacian smoothing operator is applied to the unstructured mesh to reposition nodes. The new node position is the average of the positions of its node neighbors. The computed node position increment is multiplied by the relaxation factor (which is set to a value between 0.0 and 1.0). A value of zero for the relaxation factor results in no movement of the node, and a value of unity results in movement equivalent to the entire computed increment. *Figure 29.16* (p. 1351) illustrates the new node position for a typical configuration of quadrilateral cells. The dashed line is the original mesh and the solid line is the final mesh.

Figure 29.16 Result of Smoothing Operator on Node Position



This repositioning strategy improves the skewness of the mesh, but relaxes the clustering of node points. In extreme circumstances, the present operator may create mesh lines that cross over the boundary, creating

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

negative cell volumes. This is most likely to occur near sharp or coarsely resolved convex corners, especially if you perform multiple smoothing operations with a large relaxation factor. *Figure 29.17* (p. 1352) illustrates an initial tetrahedral mesh before one unrelaxed smoothing iteration creates mesh lines that cross over each other (*Figure 29.18* (p. 1352)).

Figure 29.17 Initial Mesh Before Smoothing Operation

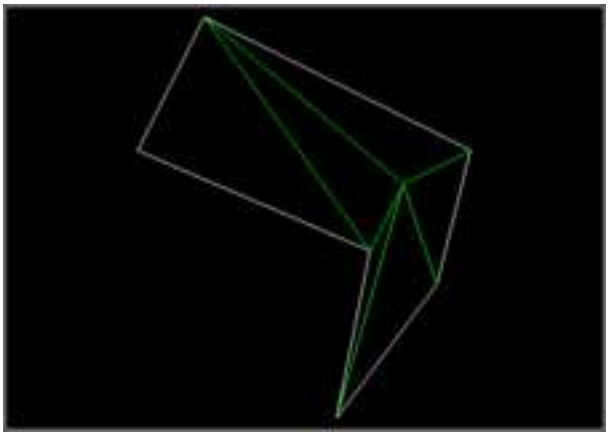
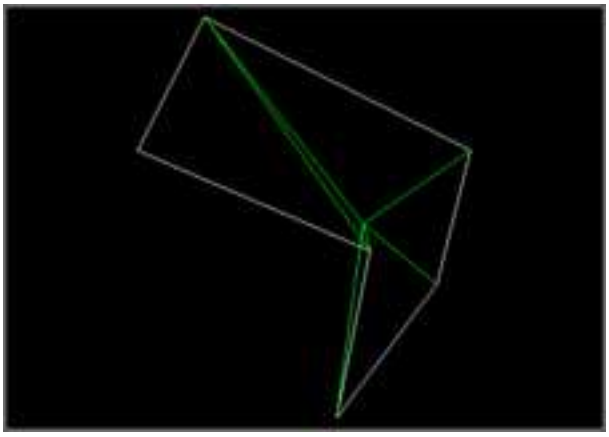


Figure 29.18 Mesh Smoothing Causing Mesh-Line Crossing



The default smoothing parameters are designed to improve mesh quality with minimal adverse effects, but it is recommended that you save a case file before smoothing the mesh. If you apply a conservative relaxation factor and start with a good quality initial mesh, the frequency of failure due to smoothing is extremely low in two dimensions. However, corruption of the mesh topology occurs much more frequently in three dimensions, particularly with tetrahedral meshes.

The smoothing operator can also be applied repeatedly, but as the number of smoothing sweeps increase, the node points have a tendency to pull away from boundaries and the mesh tends to lose any clustering characteristics.

To perform Laplacian smoothing, do the following steps:

1. In the *Smooth/Swap Mesh Dialog Box* (p. 2135) (*Figure 29.15* (p. 1350)), select **laplace** from the **Method** drop-down list in the **Smooth** group box.
2. Set the factor by which to multiply the computed position increment for the node in the **Relaxation Factor** field. The lower the factor, the more reduction in node movement.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the number of successive smoothing sweeps to be performed on the mesh in the **Number of Iterations** field. The default value is 4.
4. Click the **Smooth** button.

29.13.1.3. Skewness-Based Smoothing

When you use skewness-based smoothing, ANSYS FLUENT applies a smoothing operator to the mesh, repositioning interior nodes to lower the maximum skewness of the mesh. ANSYS FLUENT will try to move interior nodes to improve the skewness of cells with skewness greater than the specified "skewness threshold". This process can be very time consuming, so perform smoothing only on cells with high skewness.

Improved results can be obtained by smoothing the nodes several times. There are internal checks that will prevent a node from being moved if moving it causes the maximum skewness to increase, but it is common for the skewness of some cells to increase when a cell with a higher skewness is being improved. Thus, you may see the average skewness increase while the maximum skewness is decreasing.

Important

Carefully consider whether the improvements to the mesh due to a decrease in the maximum skewness are worth the potential increase in the average skewness. Performing smoothing only on cells with very high skewness (e.g., 0.8 or 0.9) may reduce the adverse effects on the average skewness.

To perform skewness-based smoothing, do the following:

1. In the *Smooth/Swap Mesh Dialog Box* (p. 2135) (*Figure 29.15* (p. 1350)), select **skewness** from the **Method** drop-down list in the **Smooth** group box.
2. Set the minimum cell skewness value for which node smoothing will be attempted in the **Skewness Threshold** field. ANSYS FLUENT will try to move interior nodes to improve the skewness of cells with skewness greater than this value. By default, **Skewness Threshold** is set to 0.4 for 2D and 0.8 for 3D.
3. Specify the number of successive smoothing sweeps to be performed on the mesh in the **Number of Iterations** field. The default value is 4.
4. Click the **Smooth** button.

29.13.2. Face Swapping

Face swapping is used to improve the quality of a triangular or tetrahedral mesh.

To perform face swapping, click the **Swap** button in the *Smooth/Swap Mesh Dialog Box* (p. 2135) until the reported **Number Swapped** is 0. The **Number Visited** indicates the total number of faces that were visited and tested for possible face swapping.

Face swapping is applicable only to meshes with triangular or tetrahedral cells.

29.13.2.1. Triangular Meshes

The approach for triangular meshes is to use the Delaunay circle test to decide if a face shared by two triangular cells should be swapped. A pair of cells sharing a face satisfies the circle test if the circumcircle of one cell does not contain the unshared node of the second cell. *Figure 29.19* (p. 1354) illustrates cell neighbors in the circle test. In cases where the circle test is not satisfied, the diagonal or face is swapped, as illustrated in *Figure 29.20* (p. 1354).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.19 Examples of Cell Configurations in the Circle Test

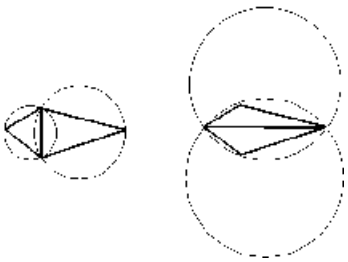


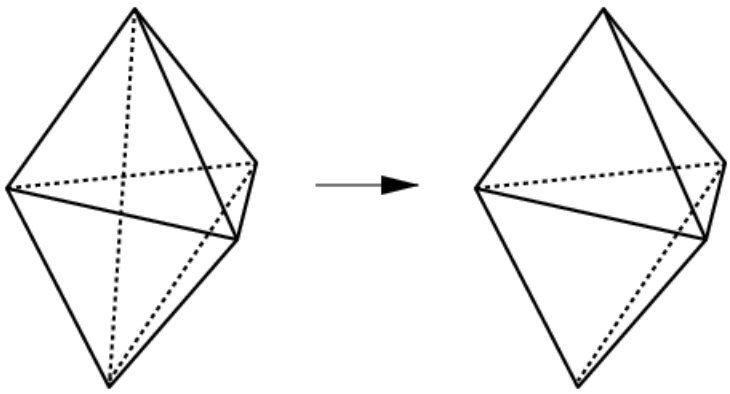
Figure 29.20 Swapped Faces to Satisfy the Delaunay Circle Test

Repeated application of the face-swapping technique will produce a constrained Delaunay mesh. If you have a Delaunay mesh, it is a unique triangulation that maximizes the minimum angles in the mesh. Thus, the triangulation tends toward equilateral cells, providing the most equilateral mesh for the given node distribution. For more information on Delaunay mesh generation, see the Theory chapter in the TGrid User's Guide.

29.13.2.2. Tetrahedral Meshes

For tetrahedral meshes, face swapping consists of searching for configurations of three cells sharing an edge and converting them into two cells sharing a face to decrease skewness and the cell count (see *Figure 29.21* (p. 1354)).

Figure 29.21 3D Face Swapping



29.13.3. Combining Skewness-Based Smoothing and Face Swapping

As mentioned in *Skewness-Based Smoothing* (p. 1353), skewness-based smoothing should usually be alternated with face swapping. Guidelines for this procedure are presented here.

- Perform four smoothing iterations using a **Skewness Threshold** of 0.8 for 3D, or 0.4 for 2D.
- Swap until the **Number Swapped** decreases to 0.
- For 3D meshes, decrease the **Skewness Threshold** to 0.6 and repeat the smoothing/swapping procedure.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 30: Creating Surfaces for Displaying and Reporting Data

ANSYS FLUENT allows you to select portions of the domain to be used for visualizing the flow field. The domain portions are called surfaces, and there are a variety of ways to create them. Surfaces are required for graphical analysis of 3D problems, since you cannot display vectors, contours, etc. or create an XY plot for the entire domain at once. In 2D you can usually visualize the flow field on the entire domain, but to create an XY plot of a variable in a portion of the interior of the domain, you must generate a surface. In addition, in both 2D and 3D, you will need one or more surfaces if you want to generate a surface-integral report. Note that ANSYS FLUENT will automatically create a surface for each boundary zone in the domain. Surface information is stored in the case file.

The following sections explain how to create surfaces, rename, group, and delete them, and determine their sizes.

30.1. Using Surfaces
30.2. Zone Surfaces
30.3. Partition Surfaces
30.4. Point Surfaces
30.5. Line and Rake Surfaces
30.6. Plane Surfaces
30.7. Quadric Surfaces
30.8. Isosurfaces
30.9. Clipping Surfaces
30.10. Transforming Surfaces
30.11. Grouping, Renaming, and Deleting Surfaces

30.1. Using Surfaces

In order to visualize the internal flow of a 3D problem or create XY plots of solution variables for 3D results, you must select portions of the domain (surfaces) on which the data is to be displayed. Surfaces can also be used for visualizing or plotting data for 2D problems, and for generating surface-integral reports.

ANSYS FLUENT provides methods for creating several kinds of surfaces, and stores all surfaces in the case file. These surfaces and their uses are described briefly below:

Zone Surfaces:

If you want to create a surface that will contain the same cells/faces as an existing cell/face zone, you can generate a zone surface. This kind of surface is useful for displaying results on boundaries.

Partition Surfaces:

When you are using the parallel version of ANSYS FLUENT, you may find it useful to create surfaces that are defined by the boundaries between mesh partitions (see *Parallel Processing* (p. 1585) for more information about running the parallel solver). You can then display data on each side of a partition boundary.

Point Surfaces:

To monitor the value of some variable or function at a particular location in the domain, you can create a surface consisting of a single point.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Line and Rake Surfaces:

To generate and display pathlines, you must specify a surface from which the particles are released. Line and rake surfaces are well-suited for this purpose and for obtaining data for comparison with wind tunnel data. A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface is simply a line that includes the specified endpoints and extends through the domain; data points will be at the centers of the cells through which the line passes, and consequently will not be equally spaced.

Plane Surfaces:

If you want to display flow-field data on a specific plane in the domain, you can create a plane surface. A plane surface is simply a plane that passes through three specified points.

Quadric Surfaces:

To display data on a line (2D), plane (3D), circle (2D), sphere (3D), or quadric surface you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces.

Isosurfaces:

You can use an isosurface to display results on cells that have a constant value for a specified variable. Generating an isosurface based on x, y, or z coordinate, for example, will give you an x, y, or z cross-section of your domain. Generating an isosurface based on pressure will allow you to display data for another variable on a surface of constant pressure.

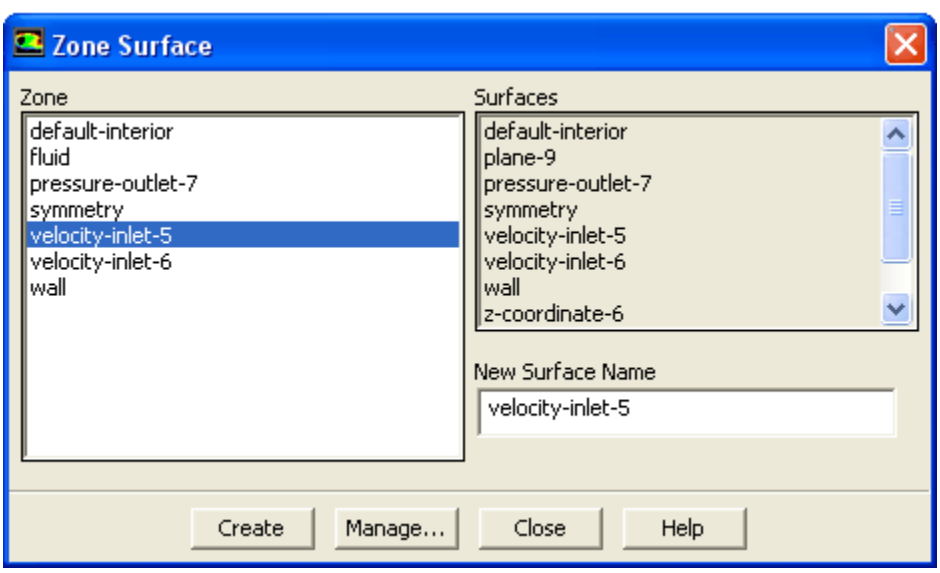
30.2. Zone Surfaces

Zone surfaces are useful for displaying results on boundaries. For example, you may want to plot contours of velocity magnitude at the inlet and outlet of the problem domain, or temperature contours on the domain's walls. To do so, you need to have a surface that contains the same faces (or cells) as an existing face (or cell) zone. Zone surfaces are created automatically for all boundary face zones in the domain, so you will generally not need to create any zone surfaces unless you accidentally delete one.

To create a zone surface, you will use the *Zone Surface Dialog Box* (p. 2137) (*Figure 30.1* (p. 1356)).

Surface → Zone...

Figure 30.1 The Zone Surface Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The steps for creating the zone surface are as follows:

1. In the **Zone** list, select the zone for which you want to create a surface.
2. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `zone-surface-6`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.

   Important

   The surface name that you enter must begin with an alphabetical letter. If your surface name begins with any other character or number, ANSYS FLUENT will not accept the entry.

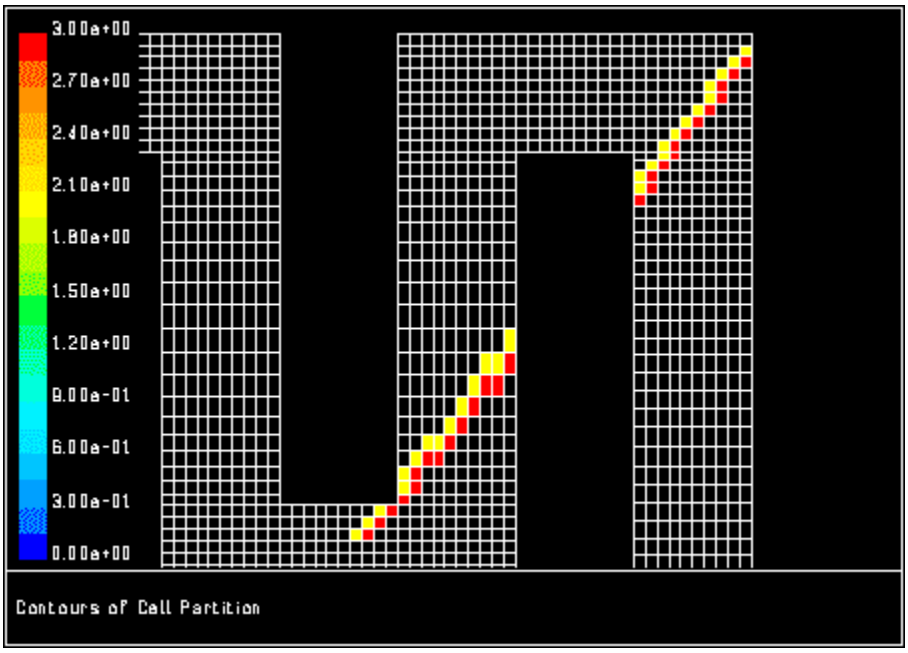
3. Click on the **Create** button. The new surface name will be added to the **Surfaces** list in the dialog box.

If you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). For details, see *Grouping, Renaming, and Deleting Surfaces* (p. 1376).

30.3. Partition Surfaces

If you are using the parallel version of ANSYS FLUENT (see *Parallel Processing* (p. 1585)), you may find it useful to create data surfaces defined by the boundaries of mesh partitions. As described in *Mesh Partitioning and Load Balancing* (p. 1603), partitioning the mesh divides it into groups of cells that can be solved on separate processors when you use a parallel solver. A partition surface will contain faces or cells on the boundary of two mesh partitions. For example, you can plot solution values on the partition surface to determine how the solution is changing across a partition interface. See *Figure 30.2* (p. 1357)

Figure 30.2 Contours of Cell Partitions on Partition Surface Overlaid on Mesh

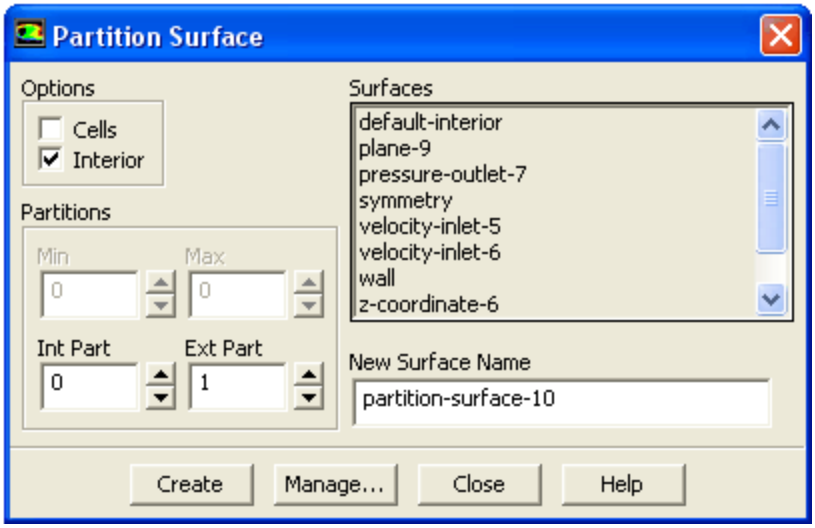


To create a partition surface, you will use the *Partition Surface Dialog Box* (p. 2137) (*Figure 30.3* (p. 1358)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface → Partition...

Figure 30.3 The Partition Surface Dialog Box



The procedure for creating the partition surface are as follows:

1. Specify the partition boundary you are interested in by indicating the two bordering partitions under the **Partitions** heading. The boundary that defines the partition surface is the boundary between the "interior partition" and the "exterior partition". **Int Part** indicates the ID number of the interior partition (i.e., the partition under consideration), and **Ext Part** indicates the ID number of the bordering (exterior) partition. The **Min** and **Max** fields will indicate the minimum and maximum ID numbers of the mesh partitions. The minimum is always zero, and the maximum is one less than the number of processors. If there are more than two mesh partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for **Int Part** and **Ext Part**, you can create surfaces for any of these boundaries.
2. Choose interior or exterior faces or cells to be contained in the partition surface by turning **Cells** and **Interior** on or off under **Options**. To obtain a surface consisting of cells that are on the "interior" side of the partition boundary, turn on both **Cells** and **Interior**. To create one consisting of cells that are on the "exterior" side, turn on **Cells** and turn off **Interior**. If you want the surface to contain the faces on the boundary instead of the cells, turn off the **Cells** option. To have the faces reflect data values for the interior cells, turn the **Interior** check button on, and to have them reflect values for the exterior cells, turn it off.
3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `partition-surface-6`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

4. Click on the **Create** button. The new surface name will be added to the **Surfaces** list in the dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). For details, see *Grouping, Renaming, and Deleting Surfaces* (p. 1376).

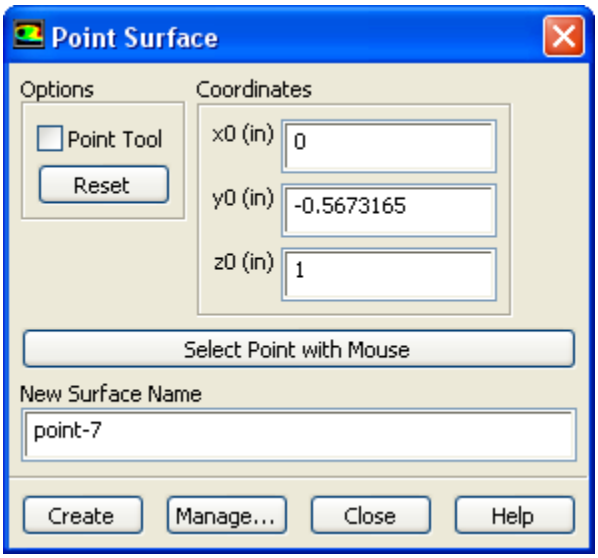
30.4. Point Surfaces

You may often be interested in displaying results at a single point in the domain. For example, you may want to monitor the value of some variable or function at a particular location. To do this, you must first create a "point" surface, consisting of a single point. When you display node-value data on a point surface, the value displayed will be a linear average of the neighboring node values. If you display cell-value data, the value at the cell in which the point lies will be displayed.

To create a point surface, you will use the *Point Surface Dialog Box* (p. 1927) (*Figure 30.4* (p. 1359)).

Surface → **Point...**

Figure 30.4 The Point Surface Dialog Box



The steps for creating the point surface are as follows:

1. Specify the location of the point. There are three different ways to do this:
   - Enter the coordinates (**x0, y0, z0**) under **Coordinates**.
   - Click on the **Select Point With Mouse** button and then select the point by clicking on a location in the active graphics window with the mouse-probe button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about setting mouse button functions.)
   - Use the **Point Tool** option to interactively position a point in the graphics window. You can set the initial location of this point using one of the two methods described above for specifying the point's position (or you can start from the position defined by the default **Coordinates**). See *Using the Point Tool* (p. 1360) for information about using the point tool.
2. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `point-5`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

3. Click on the **Create** button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). For details, see *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

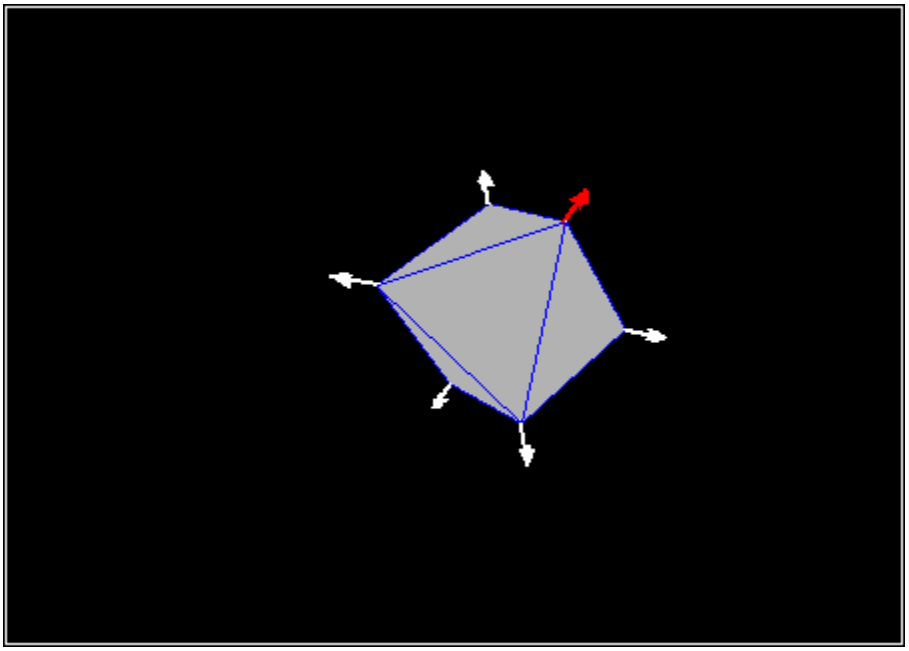
30.4.1. Using the Point Tool

The point tool allows you to interactively fine-tune the definition of a point using graphics. Starting from an initial point, you can translate the point until its position is as desired. For example, if you need to position a point surface at the center of a duct, just past the inlet, you can start with the point tool near the desired location (e.g., on the inlet), and translate it until it is in the proper place. (You may find it helpful to display mesh faces to ensure that the point tool is correctly positioned inside the domain.)

30.4.1.1. Initializing the Point Tool

Before turning on the **Point Tool** option, set the **Coordinates** to suitable starting values. You can enter values manually, or use the **Select Point With Mouse** button. Often it is convenient to display the mesh for an inlet or isosurface on or near which the point is to be located, and then select a point on that mesh to specify the initial position of the point tool. Once you have specified the appropriate **Coordinates**, activate the tool by turning on the **Point Tool** option. The point tool, an eight-sided polygon, will appear in the graphics window, as shown in *Figure 30.5* (p. 1360).

Figure 30.5 The Point Tool



You can then translate the point tool as described below. The point surface you create will be located at the center of the point tool.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.4.1.2. Translating the Point Tool

To translate the point tool in the direction along the red axis, click the mouse-probe button (the right button by default) anywhere on the gray part of the point tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion. For information about changing the mouse functions *Controlling the Mouse Button Functions* (p. 1427).

To translate the tool in the transverse directions (i.e., along either of the other axes), press the `<Shift>` key, click the mouse-probe button anywhere on the gray part of the point tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. In 2D, there will be only one set of green arrows, since there is only one other direction for translation. If you find the perspective distracting when performing this type of translation, you can turn it off in the *Camera Parameters Dialog Box* (p. 2005) (opened from the *Views Dialog Box* (p. 2000)), as described in *Controlling Perspective and Camera Parameters* (p. 1438).

30.4.1.3. Resetting the Point Tool

If you "lose" the point tool, or want to reset it for any other reason, you can either click on the **Reset** button to return the point tool to the default position and start from there, or turn the tool off and reinitialize it as described above. In the default position, the point tool will lie at the center of the domain.

30.5. Line and Rake Surfaces

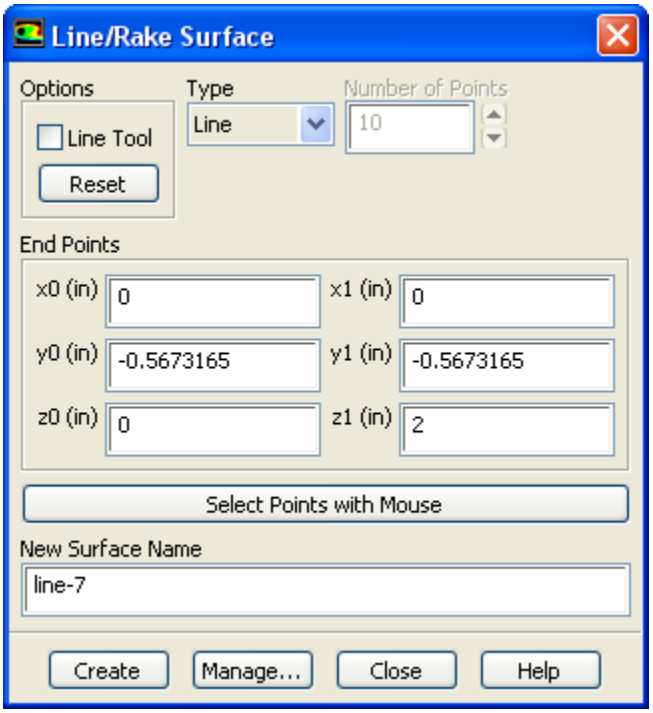
You can create lines and rakes in the domain for releasing particles, obtaining data for comparison with tunnel data, etc. A rake consists of a specified number of points equally spaced between two specified end-points. A line is simply a line that extends up to and includes the specified endpoints; data points will be located where the line intersects the faces of the cell, and consequently may not be equally spaced.

To create a line or rake surface, you will use the *Line/Rake Surface Dialog Box* (p. 1928) (*Figure 30.6* (p. 1362)).

Surface → **Line/Rake...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.6 The Line/Rake Surface Dialog Box



The steps for creating the line or rake surface are as follows:

1. Indicate whether you are creating a **Line** surface or a **Rake** surface by selecting the appropriate item in the **Type** drop-down list.
2. If you are creating a rake surface, specify the **Number of Points** to be equally spaced between the two endpoints.
3. Specify the location of the line or rake surface. There are three different ways to define the location:
   - Enter the coordinates of the first point (**x0, y0, z0**) and the last point (**x1, y1, z1**) under **End Points**.
   - Click on the **Select Points With Mouse** button and then select the endpoints by clicking on locations in the active graphics window with the mouse-probe button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about setting mouse button functions.)
   - Use the **Line Tool** option to interactively position a line in the graphics window. You can set the initial location of this line using one of the two methods described above for specifying endpoints (or you can start from the position defined by the default **End Points**). See *Using the Line Tool* (p. 1363) for information about using the line tool.

   Note that the coordinates of the **End Points** will be updated automatically when you use the second or third method described above.
4. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `line-5` or `rake-6`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

5. Click on the **Create** button to create the new surface.

If you want to check that the new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

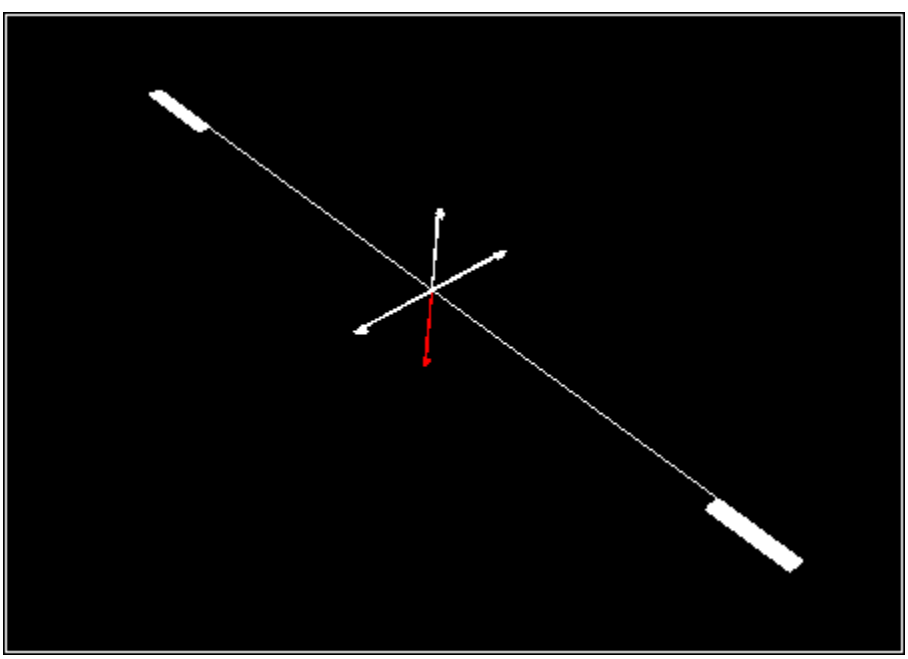
30.5.1. Using the Line Tool

The line tool allows you to interactively fine-tune the definition of a line or rake using graphics. Starting from an initial line, you can translate, rotate, and resize the line until its position, orientation, and length are as desired. For example, if you need to position a rake surface just inside the inlet to a duct, you can start with the line tool near the desired location (e.g., on the inlet), and translate, rotate, and resize it until you are satisfied. You may find it helpful to display mesh faces to ensure that the line tool is correctly positioned inside the domain.

30.5.1.1. Initializing the Line Tool

Before turning on the **Line Tool** option, set the **End Points** to suitable starting values. You can enter values manually, or use the **Select Points With Mouse** button. Often it is convenient to display the mesh for an inlet or isosurface on or near which you wish to place the line or rake surface and then select two points on that mesh to specify the initial position of the line tool. Once you have specified the appropriate **End Points**, activate the tool by turning on the **Line Tool** option. The line tool will appear in the graphics window, as shown in *Figure 30.7* (p. 1363).

Figure 30.7 The Line Tool



You can then translate, rotate, and/or resize the line tool as described in the following sections.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.5.1.2. Translating the Line Tool

To translate the line tool in the direction along the red axis, click the mouse-probe button (the right button by default) anywhere on the "line" part of the tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion For information about changing the mouse functions see *Controlling the Mouse Button Functions* (p. 1427).

Important

Do not click on the axes of the line tool that have arrows on the ends. These axes control rotation of the tool. Click only on the portion of the tool that represents the prospective line surface. This portion is designated by the rectangles attached to each end.

To translate the tool in the transverse directions (i.e., along either of the axes within the plane perpendicular to the red axis), press the `<Shift>` key, click the mouse-probe button anywhere on the "line" part of the tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. In 2D, there will be only one set of green arrows, since there is only one other direction for translation. If you find the perspective distracting when performing this type of translation, you can turn it off in the *Camera Parameters Dialog Box* (p. 2005) (opened from the *Views Dialog Box* (p. 2000)), as described in *Controlling Perspective and Camera Parameters* (p. 1438).

30.5.1.3. Rotating the Line Tool

To rotate the line tool, you will click the mouse-probe button on one of the white axes with arrows. When you click on one of these axes, a green ribbon will encircle the other arrowed axis, designating it as the axis of rotation. As you drag the mouse along the circle to rotate the tool, the green circle will become yellow.

Important

Do not click on the red axis to rotate the line tool.

30.5.1.4. Resizing the Line Tool

If you plan to generate a rake surface, you can resize the line tool to define the length of the rake. Click the mouse-probe button in one of the white rectangles at the ends of the "line" part of the tool (shown in black in *Figure 30.7* (p. 1363)) and drag the mouse to lengthen or shorten the tool. Green arrows will show the direction of stretching/shrinking.

30.5.1.5. Resetting the Line Tool

If you "lose" the line tool, or want to reset it for any other reason, you can either click on the **Reset** button to return the line tool to the default position and start from there, or turn the tool off and reinitialize it as described above. In the default position, the line tool will lie midway along the x and y lengths of the domain, spanning the z domain extent.

30.6. Plane Surfaces

To display flow-field data on a specific plane in the domain, you will use a plane surface. You can create surfaces that cut through the solution domain along arbitrary planes only in 3D; this feature is not available in 2D.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are six types of plane surfaces that you can create:

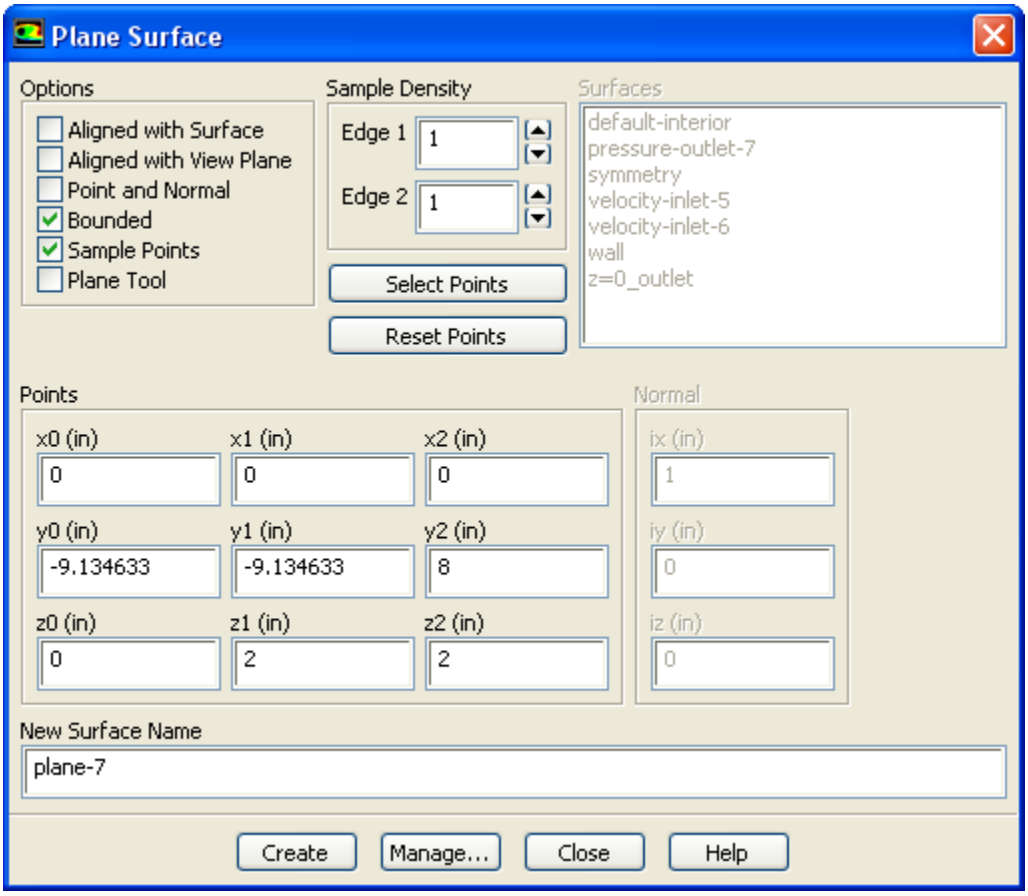
- Intersection of the domain with the infinite plane: This is the default plane surface created. The extents of the plane will be determined by the extents of the domain. Since the plane is slicing through the domain, the data points will, by default, be located where the plane intersects the faces of a cell, and consequently may not be equally spaced.
- Bounded plane: This plane will be a bounded parallelepiped, for which 3 of the 4 corners are the 3 points that define the plane equation (or the 4 corners are the corners of the "plane tool"). Like the default plane surface described above, this type of surface will also have unequally spaced data points.
- Bounded plane with equally spaced data points: This plane is the same as the bounded plane described above, except you will specify the density of points along the two directions of the parallelepiped, creating a uniform distribution of data points.
- Plane having a certain normal vector and passing through a specified point: To create this type of plane, you will define a normal vector and a point. A plane with the specified normal and passing through the specified point will be created.
- Plane aligned with an existing surface: To create this type of plane, you will define a single point and a surface. A plane parallel to the selected surface and passing through the specified point will be created.
- Plane aligned with the view in the graphics window: To create this type of plane, you will define a single point. A plane parallel to the current view in the active graphics window and passing through the specified point will be created.

To create a plane surface, you will use the *Plane Surface Dialog Box* (p. 1929) (*Figure 30.8* (p. 1366)).

Surface → **Plane...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.8 The Plane Surface Dialog Box



The procedure for creating the plane surface is as follows:

1. Decide which of the six types of planes you want to create.
   - To create the default plane type (the intersection of the infinite plane with the domain), go directly to step 2.
   - To create a bounded plane, turn on **Bounded** under **Options**.
   - To create a bounded plane with equally spaced data points, turn on both **Bounded** and **Sample Points**, and then set the number of data points under **Sample Density**. You will specify the point density in each direction by entering the appropriate values for **Edge 1** and **Edge 2**. Edge 1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2.
   - To define a plane aligned with an existing surface, select **Aligned With Surface**, and then choose the surface in the **Surfaces** list and specify a single point using one of the first two methods described below in step 2.
   - To define a plane aligned with the view plane, select **Aligned With View Plane**, and then choose a single point using one of the first two methods described below in step 2.
   - To define a plane having a certain normal vector and passing through a specified point, select **Point And Normal**, and then specify the normal vector by entering values in the **ix**, **iy**, and **iz** fields under **Normal**, and a single point using one of the first two methods described below in step 2.
2. Specify the location of the plane surface. There are three different ways to define the location:
   - Enter the coordinates of the three **Points** defining the planar surface: (**x0, y0, z0**), (**x1, y1, z1**),and (**x2, y2, z2**).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click on the **Select Points** button and then select the three points by clicking on locations in the active graphics window with the mouse-probe button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about setting mouse button functions.)
- Use the **Plane Tool** option to interactively position a plane in the graphics window. You can set the initial location of this plane using one of the two methods described above for specifying the defining points. You can also start from the position defined by the default **Points**. See *Using the Plane Tool* (p. 1367) for information about using the plane tool.

Note that the coordinates of the **Points** will be updated automatically when you use the second or third method described above.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `plane-7`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

4. Click on the **Create** button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

30.6.1. Using the Plane Tool

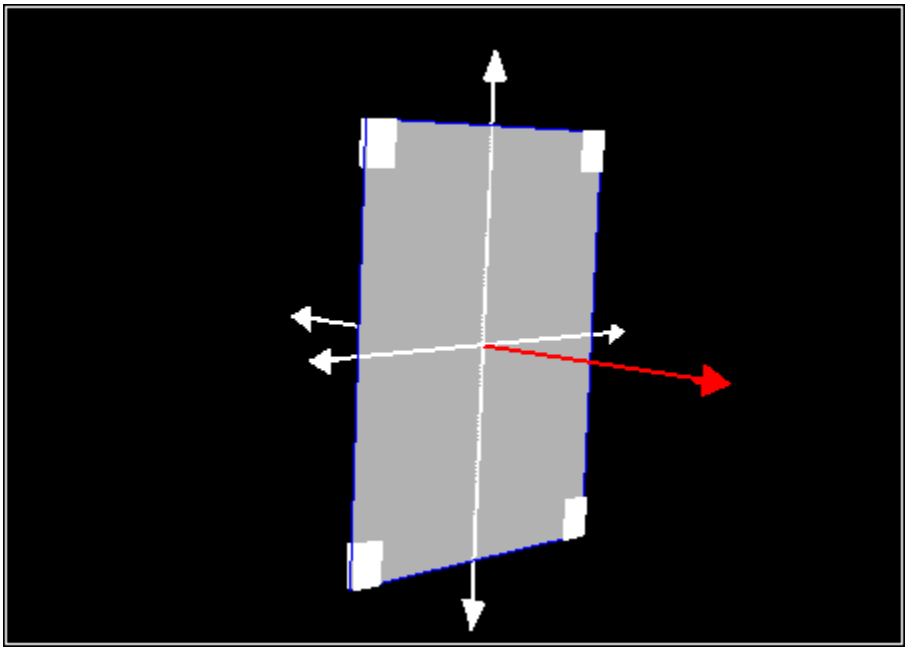
The plane tool allows you to interactively fine-tune the definition of a plane using graphics. Starting from an initial plane, you can translate, rotate, and resize the plane until its position, orientation, and size are as desired. For example, if you need to position a plane surface at a cross-section of an irregularly-shaped, curved duct, you can start with the plane tool near the desired location, resize it, translate it until it is within the duct walls, and rotate it to the proper orientation. You may find it helpful to display mesh faces to ensure that the plane tool is correctly positioned inside the domain.

30.6.1.1. Initializing the Plane Tool

Before turning on the **Plane Tool** option, set the **Points** to suitable starting values. You can enter values manually, or use the **Select Points** button. Often it is convenient to display the mesh for an inlet or isosurface that is similar to the desired plane surface, and then select three points on that mesh to position the initial plane. Once you have specified the appropriate **Points**, activate the tool by turning on the **Plane Tool** option. The plane tool will appear in the graphics window, as shown in *Figure 30.9* (p. 1368).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.9 The Plane Tool



You can then translate, rotate, and/or resize the plane tool as described in the following sections.

30.6.1.2. Translating the Plane Tool

To translate the plane tool in the direction normal to the plane, click the mouse-probe button (the right button by default) anywhere on the gray part of the plane tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion. For information about changing the mouse functions see *Controlling the Mouse Button Functions* (p. 1427).

To translate the tool in the transverse directions (i.e., along either of the axes that lie within the plane), press the `<Shift>` key, click the mouse-probe button anywhere on the gray part of the plane tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. If you find the perspective distracting when performing this type of translation, you can turn it off in the *Camera Parameters Dialog Box* (p. 2005) (opened from the *Views Dialog Box* (p. 2000)), as described in *Controlling Perspective and Camera Parameters* (p. 1438).

30.6.1.3. Rotating the Plane Tool

To rotate the plane tool, you will click the mouse-probe button on one of the white arrows at the tips of the plane's axes. Clicking on any arrow allows you to rotate the tool about either of the other two axes: when you click on the arrow, two green ribbons will encircle the plane tool, forming circles about each of the two possible axes of rotation. Drag the mouse along the desired circle to rotate the tool. As you do so, the circle along which the tool is rotating will become yellow.

The following notes may help you when you are rotating the plane tool:

- Once you move your mouse along one circle, you cannot change the direction of rotation unless you release the mouse-probe button and try again. Be careful to start moving your mouse very steadily so that you can choose the correct direction.
- Do not click on the red arrow to rotate.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Do not try to rotate by clicking on an arrow that is pointing away from you. It will be very difficult for you to judge which direction of rotation is correct from this point of view. Since there are two arrows on each axis, there will always be an appropriate arrow available.
- Do not rotate the plane tool more than 90° or so at once. If you rotate the tool by a large angle, the arrow on which you are clicking will begin to point away from you, and you will have trouble controlling the rotation (as discussed in the item above).

30.6.1.4. Resizing the Plane Tool

If you plan to generate a bounded plane, you can resize the plane tool to define the plane's boundaries. Click the mouse-probe button in one of the white squares at the plane tool's corners (shown in black in *Figure 30.9* (p. 1368)) and drag the mouse to stretch or shrink the tool. Green arrows will show the direction of the plane's diagonal.

Important

Be careful not to drag your mouse across any of the axes while resizing the tool. This will flip the tool over and corrupt it. If you accidentally do this, reset the plane tool and start again.

30.6.1.5. Resetting the Plane Tool

If you "lose" the plane tool, or want to reset it for any other reason, you can either click on the **Reset Points** button to return the plane tool to the default position and start from there, or turn the tool off and reinitialize it as described above. In the default position, the plane tool will lie midway along the x length of the domain, spanning the y and z domain extents.

30.7. Quadric Surfaces

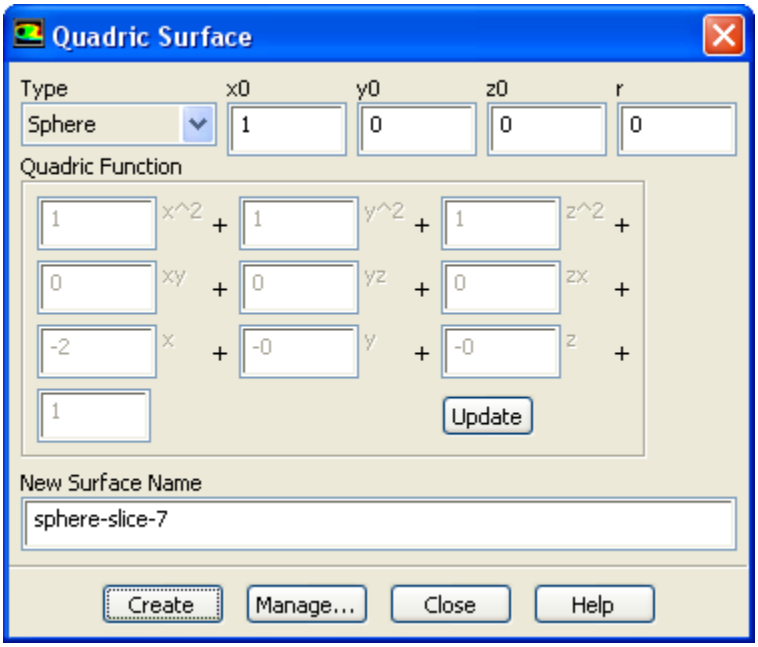
If you want to display data on a line (2D), plane (3D), circle (2D), sphere (3D), or general quadric surface, you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces. See *Line and Rake Surfaces* (p. 1361) and *Plane Surfaces* (p. 1364) for additional methods for creating line and plane surfaces.

To create a quadric surface, you will use the *Quadric Surface Dialog Box* (p. 1931) (*Figure 30.10* (p. 1370)).

Surface → **Quadric...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.10 The Quadric Surface Dialog Box



The steps for creating the quadric surface are as follows:

1. Decide which type of quadric surface you want to create. In 3D, choose **Plane**, **Sphere**, or (general) **Quadric** in the **Type** drop-down list. In 2D, choose **Line**, **Circle**, or **Quadric**.
2. Specify the defining equation for the surface *in SI units*.
   - Line or plane surface: If you have selected **Line** (in 2D) or **Plane** (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=\text{distance}$. You will input **ix** (the coefficient of x), **iy** (the coefficient of y), **iz** (the coefficient of z), and **distance** (the distance of the line or plane from the origin) in the fields to the right of the **Type** drop-down list. When you click on the **Update** button under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.
   - Circle or sphere surface: If you have selected **Circle** (in 2D) or **Sphere** (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$. You will input **x0, y0, z0** (the x, y, and z coordinates of the sphere or circle's center) and **r** (the radius) in the fields to the right of the **Type** drop-down list. When you click on the **Update** button under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.
   - Quadric surface: If you have selected **Quadric** as the surface type, the surface will consist of all points in the domain that satisfy the general quadric function $Q=\text{value}$. You will input the coefficients of the quadric function Q (the coefficients of the terms x^2, y^2, z^2, xy, yz, zx, x, y, z and the constant term) directly in the **Quadric Function** box, and you will set **value** to the right of the **Type** drop-down list. Note that the **Update** button will be disabled when you choose this type of surface.
3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `sphere-slice-7` or `quadric-slice-10`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

4. Click on the **Create** button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

30.8. Isosurfaces

If you want to display results on cells that have a constant value for a specified variable, you will need to create an isosurface of that variable. Generating an isosurface based on x, y, or z coordinate, for example, will give you an x, y, or z cross-section of your domain; generating an isosurface based on pressure will allow you to display data for another variable on a surface of constant pressure. You can create an isosurface from an existing surface or from the entire domain. Furthermore, you can restrict any isosurface to a specified cell zone.

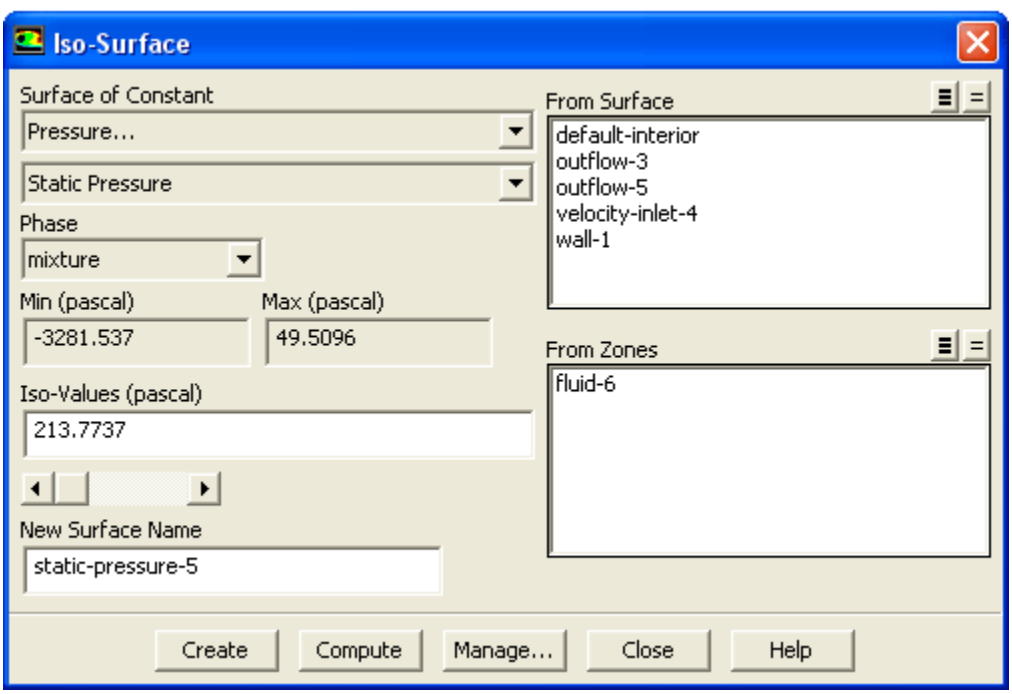
Important

Note that you cannot create an isosurface until you have initialized the solution, performed calculations, or read a data file.

To create an isosurface, you will use the *Iso-Surface Dialog Box* (p. 1933) (*Figure 30.11* (p. 1371)).

Surface → **Iso-Surface...**

Figure 30.11 The Iso-Surface Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The steps for creating the isosurface are as follows:

1. Choose the scalar variable to be used for isosurfacing in the **Surface of Constant** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
2. If you wish to create an isosurface from an existing surface (i.e., generate a new surface of constant x, y, temperature, pressure, etc. that is a subset of another surface), choose that surface in the **From Surface** list. You can specify the cell zone on which you want to create an isosurface by selecting the zone in the **From Zones** list.
   - If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.
   - If you do not select a zone from the list, then the isosurfacing will not be restricted to any cell zone and will run through the entire domain.
3. Click on the **Compute** button to calculate the minimum and maximum values of the selected scalar field in the domain or on the selected surface (in the **From Surface** list). The minimum and maximum values will be displayed in the **Min** and **Max** fields.
4. Set the isovalue using one of the following methods. (Note that the second method will allow you to define multiple isovalues in a single isosurface.)
   - You can set an isovalue interactively by moving the slider with the left mouse button. The value in the **Iso-Values** field will be updated automatically. This method will also create a temporary isosurface in the graphics window. Using the slider allows you to preview an isosurface before creating it.

     Important

     Even though the isosurface is displayed, it is only a temporary surface. To create an isosurface, use the **Create** button after deciding on a particular isovalue.

   - You can type in isovalues in the **Iso-Values** field directly, separating multiple values by white space. Multiple isovalues will be contained in a single isosurface; i.e., you cannot select subsurfaces within the resulting isosurface.
5. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `z-coordinate-6`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

6. Click on the **Create** button. The new surface name will be added to the **From Surface** list in the dialog box.

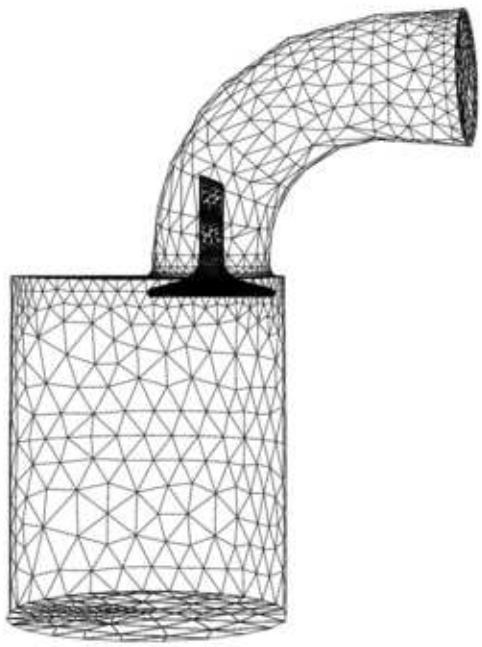
If you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.9. Clipping Surfaces

If you have created a surface, but you do not want to use the whole surface to display data, you can clip the surface between two isovalues to create a new surface that spans a specified subrange of a specified scalar quantity. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. For example, in *Figure 30.12* (p. 1373) the external wall has been clipped to values of x coordinate less than 0 to show only the back half of the wall, allowing you to see the valve inside the intake port.

Figure 30.12 External Wall Surface Isoclipped to Values of x Coordinate

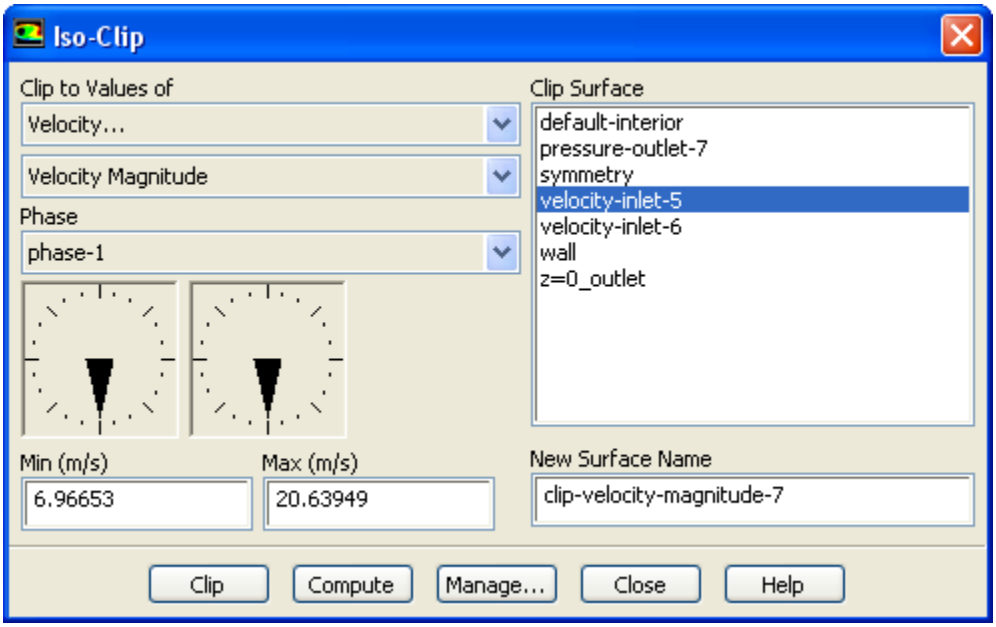


To clip an existing surface, you will use the *Iso-Clip Dialog Box* (p. 1935) (*Figure 30.13* (p. 1374)).

Surface $\rightarrow$ **Iso-Clip...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.13 The Iso-Clip Dialog Box



The steps for clipping a surface are as follows:

1. Choose the scalar variable on which the clipping will be based in the **Clip To Values Of** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.
2. Select the surface to be clipped in the **Clip Surface** list.
3. Click on the **Compute** button to calculate the minimum and maximum values of the selected scalar field on the selected surface. The minimum and maximum values will be displayed in the **Min** and **Max** fields.
4. Define the clipping range using one of the following methods.
   - You can set the upper and lower limits of the clipping range interactively by moving the indicator in each dial (i.e., the dial above the **Min** or **Max** field) with the left mouse button. The value in the corresponding **Min** or **Max** field will be updated automatically. This method will also create a temporary surface in the graphics window. Using the dials allows you to preview a clipped surface before creating it.

     Important

     Even though the clipped surface is displayed, it is only a temporary surface. To create the new surface, use the **Clip** button after deciding on the clipping range.

   - You can type the minimum and maximum values in the clipping range directly in the **Min** and **Max** fields.
5. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `clip-density-8`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

6. Click on the **Clip** button. The new surface name will be added to the **Clip Surface** list in the dialog box. (The original surface will remain unchanged.)

If you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

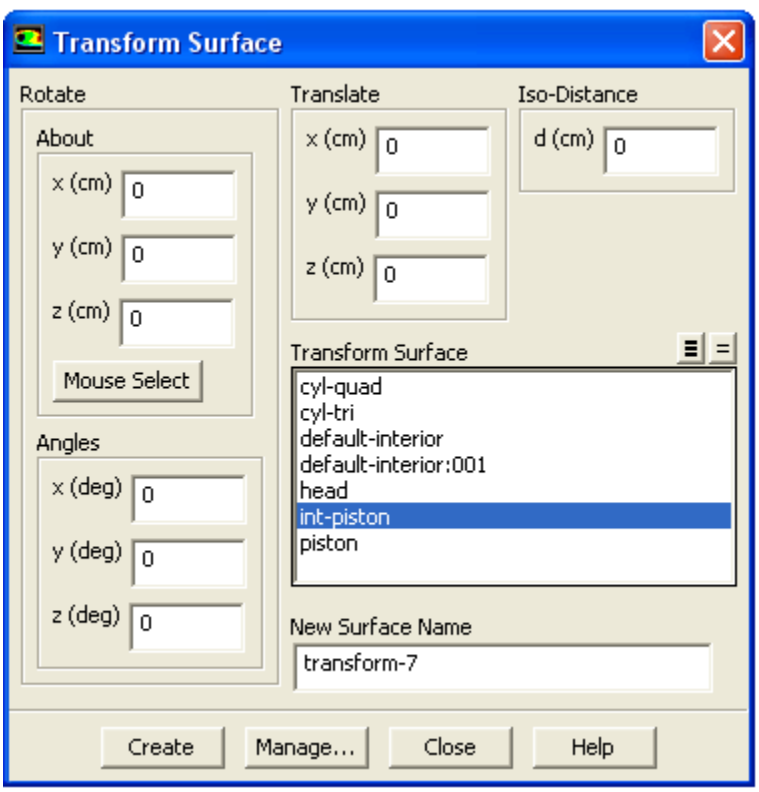
30.10. Transforming Surfaces

You can create a new data surface from an existing surface by rotating and/or translating the original surface. For example, you can rotate the surface of a complicated turbomachinery blade to plot data in the region between blades. You can also create a new surface at a constant normal distance from the original surface.

To transform an existing surface to create a new one, you will use the *Transform Surface Dialog Box* (p. 2139) (*Figure 30.14* (p. 1375)).

Surface → **Transform...**

Figure 30.14 The Transform Surface Dialog Box



The steps for transforming a surface are as follows:

1. Select the surface to be transformed in the **Transform Surface** list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the appropriate transformation parameters, as described below. You can perform any combination of translation, rotation, and "isodistancing" on the surface.
   - Rotation: To rotate a surface, you will specify the origin about which the rotation is performed, and the angle by which the surface is rotated.

     In the **About** box under **Rotate**, you will specify a point, and the origin of the coordinate system for the rotation will be set to that point. (The x, y, and z directions will be the same as for the global coordinate system.) For example, if you specified the point (1,5,3) in 3D, rotation would be about the x, y, and z axes anchored at (1,5,3). You can either enter the point's coordinates in the **x,y,z** fields or click on the **Mouse Select** button and select a point in the graphics window using the mouse-probe button. See *Controlling the Mouse Button Functions* (p. 1427) for information about mouse button functions.

     In the **Angles** box under **Rotate**, you will specify the angles about the x, y, and z axes (i.e., the axes of the coordinate system with the origin defined under **About**) by which the surface is rotated. For 2D problems, you can specify rotation about the z axis only.
   - Translation: To translate a surface, you will simply define the distance by which the surface is translated in each direction. Set the **x**, **y**, and **z** translation distances under **Translate**.
   - Isodistancing: To create a surface positioned at a constant normal distance from the original surface, you need to set only that normal distance between the original surface and the transformed surface. Set the value for **d** under **Iso-Distance**.
3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., `transform-9`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS FLUENT will automatically assign the default name to the new surface when it is created.

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

4. Click on the **Create** button. The new surface name will be added to the **Transform Surface** list in the dialog box. The original surface will remain unchanged.

If you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the *Surfaces Dialog Box* (p. 1936). See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details.

30.11. Grouping, Renaming, and Deleting Surfaces

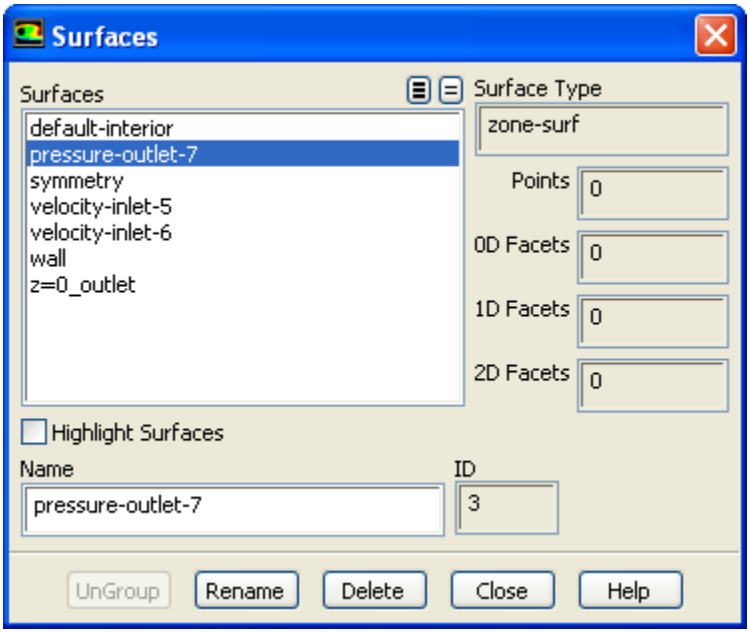
Once you have created a number of surfaces, you can interactively rename, delete, and group surfaces and obtain information about their components. Grouping surfaces is useful if you want to perform postprocessing on a number of surfaces at a time. For example, you may want to group several wall surfaces together to generate a contour plot of temperature on all walls. To postprocess results on each wall surface individually, "ungroup" the surfaces.

Manipulation of existing surfaces is performed with the *Surfaces Dialog Box* (p. 1936) (*Figure 30.15* (p. 1377)).

Surface → **Manage...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.15 The Surfaces Dialog Box



You can also open this dialog box by clicking on the **Manage...** button in one of the surface creation dialog boxes described in the previous sections.

For additional information, please see the following sections:

30.11.1. Grouping Surfaces
30.11.2. Renaming Surfaces
30.11.3. Deleting Surfaces
30.11.4. Surface Statistics

30.11.1. Grouping Surfaces

As mentioned above, you may want to group several surfaces together in order to perform postprocessing on all of them at once. To create a surface group, select the surfaces to be grouped in the **Surfaces** list. You can define a new name for the group in the **Name** field, or you can use the default name, which is the name of the first surface you selected in the **Surfaces** list. Then click on the **Group** button. The selected surfaces will disappear from the **Surfaces** list, and the name of the surface group will be added to the list.

Important

The **Group** button will not appear until you have selected at least two surfaces. As soon as you choose a second surface in the **Surfaces** list, the **Rename** button will change to the **Group** button.

To ungroup the surfaces, simply select the surface group in the **Surfaces** list and click on the **UnGroup** button. The group name will disappear from the list and the names of the original surfaces in the group will reappear in the list.

30.11.2. Renaming Surfaces

To change the name of an existing surface, select the surface in the **Surfaces** list, enter a new name in the **Name** field, and then click on the **Rename** button. The new name will replace the old name in the **Surfaces** list and the surface will be otherwise unchanged.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have selected more than one surface, the **Rename** button will not appear in the dialog box. When more than one surface is selected, the **Rename** button is replaced by the **Group** button.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS FLUENT will not accept the entry.

30.11.3. Deleting Surfaces

If you find that a surface is no longer useful, you may want to delete it to prevent the list of surfaces from becoming too cluttered. Select the surface or surfaces to be deleted in the **Surfaces** list, and then click on the **Delete** button. The delete operation is not reversible, so if you want to get a deleted surface back again you will need to recreate it using one of the surface-creation dialog boxes described in the previous sections.

30.11.4. Surface Statistics

You can also use the *Surfaces Dialog Box* (p. 1936) to retrieve topological information about surfaces. **Points** is the total number of nodes in a surface. **0D Facets** is the number of isolated nodes in a surface (i.e., nodes that have no connectivity, such as point surfaces or nodes in a rake), **1D Facets** is the number of linear faces (consisting of two connected nodes) in a surface in a 2D problem, and **2D Facets** is the number of 2D faces (triangular or quadrilateral) in a surface in a 3D problem. Note that an **interior** zone surface in a 3D problem consists of 2D facets, and similarly an **interior** zone surface in a 2D problem consists of 1D facets.

These statistics are listed for the surface(s) selected in the **Surfaces** list. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 31: Displaying Graphics

Graphics tools available in ANSYS FLUENT allow you to process the information contained in your CFD solution and easily view the results. The following sections explain how to use these tools to examine your solution. The procedure for saving picture files of graphics displays is described in *Saving Picture Files* (p. 117).

31.1. Basic Graphics Generation
31.2. Customizing the Graphics Display
31.3. Controlling the Mouse Button Functions
31.4. Viewing the Application Window
31.5. Modifying the View
31.6. Composing a Scene
31.7. Animating Graphics
31.8. Creating Videos
31.9. Histogram and XY Plots
31.10. Turbomachinery Postprocessing
31.11. Fast Fourier Transform (FFT) Postprocessing

31.1. Basic Graphics Generation

In ANSYS FLUENT you can generate graphics displays showing meshes, contours, profiles, vectors, and pathlines. Some graphics are generated using variables that are plotted directly from the ANSYS FLUENT data file once the file has been read. The variables listed in the data file depend on the models active at the time the file is written. Variables that are required by the solver, based on the current model settings, but are missing from the data file are set to their default values. For those missing variables, one iteration should be performed in order to obtain the required values for generating the plot. A complete list of variables stored in the data file is available in (`xfile.h`) and can be accessed as stated in *Data Sections* (p. 2201). The following sections describe how to create these plots. (Generation of histogram and XY plots is discussed in *Histogram and XY Plots* (p. 1466).)

Important

If your model includes a discrete phase, you can also display the particle trajectories, as described in *Displaying of Trajectories* (p. 1068).

31.1.1. Displaying the Mesh
31.1.2. Displaying Contours and Profiles
31.1.3. Displaying Vectors
31.1.4. Displaying Pathlines
31.1.5. Displaying Results on a Sweep Surface
31.1.6. Hiding the Graphics Window Display

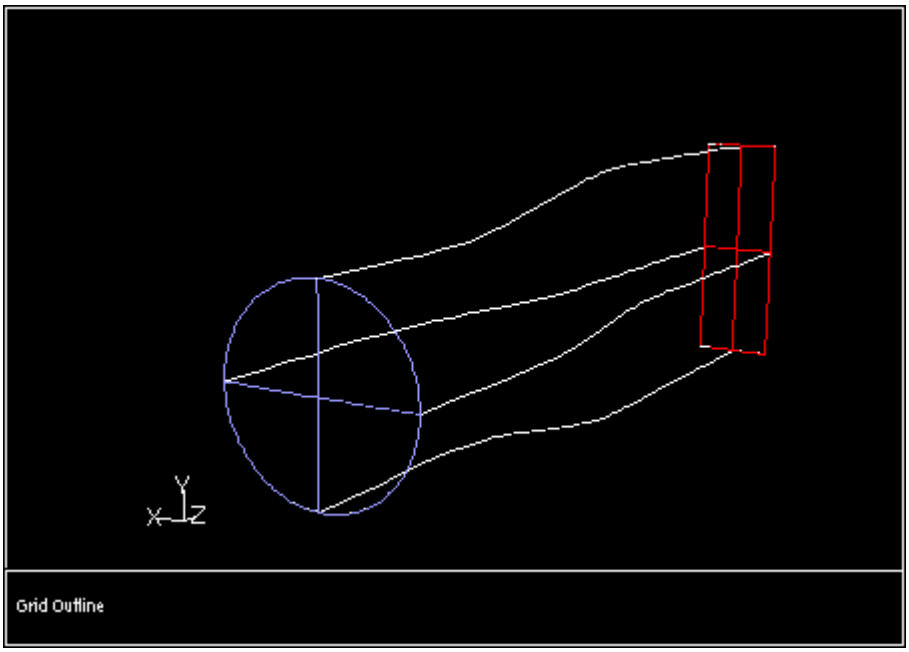
31.1.1. Displaying the Mesh

During the problem setup or when you are examining your solution, you may want to look at the mesh associated with certain surfaces. You can do the following:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

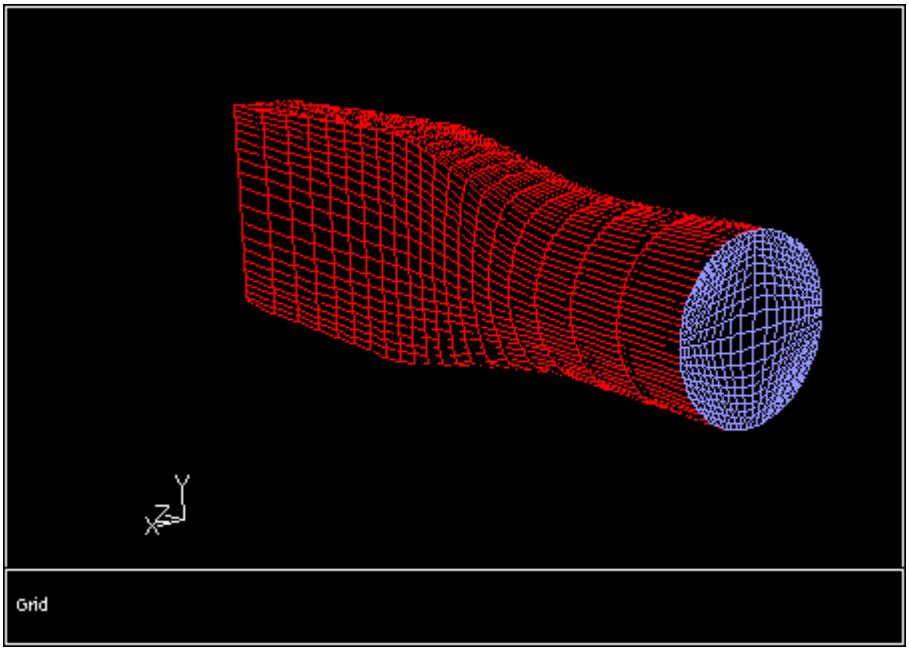
- Display the outline of all or part of the domain, as shown in *Figure 31.1* (p. 1380).

Figure 31.1 Outline Display



- Draw the mesh lines (edges), as shown in *Figure 31.2* (p. 1380).

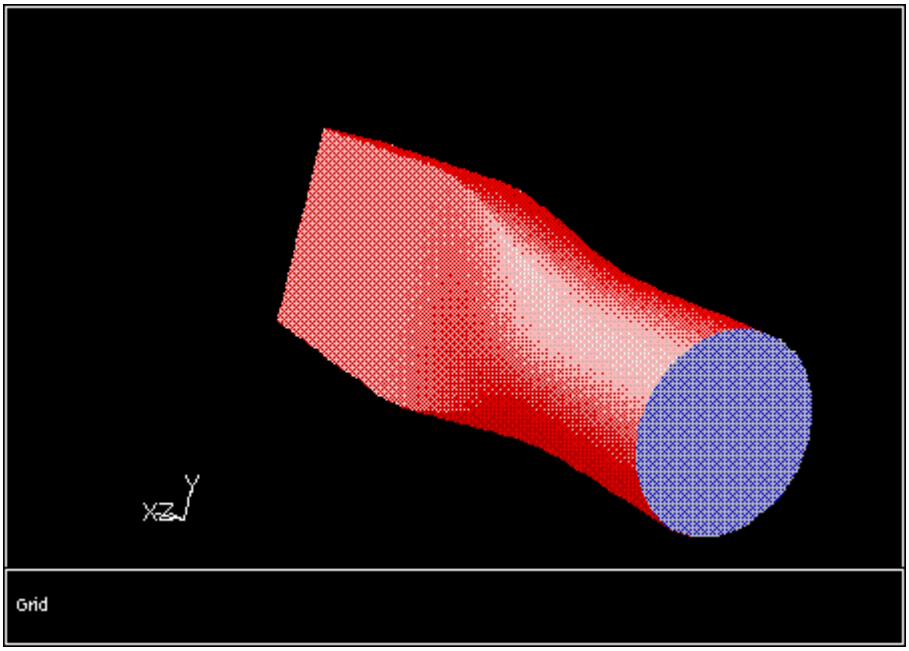
Figure 31.2 Mesh Edge Display



- Draw the solid surfaces (filled meshes) for a 3D domain, as shown in *Figure 31.3* (p. 1381).

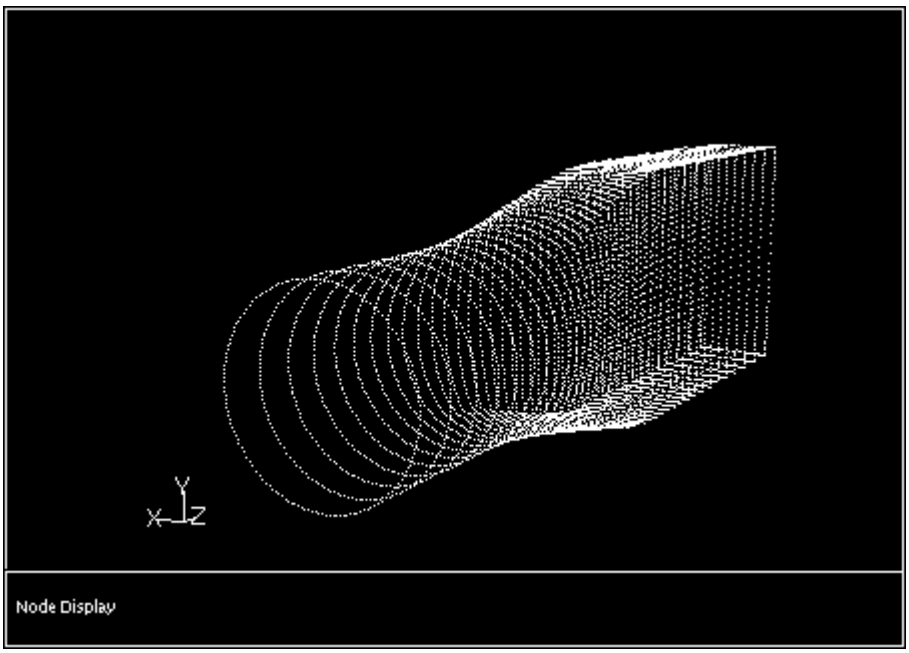
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.3 Mesh Face (Filled Mesh) Display



- Draw the nodes on the domain surfaces, as shown in *Figure 31.4* (p. 1381).

Figure 31.4 Node Display



For information about displaying the mesh on a surface that sweeps through the domain, see *Displaying Results on a Sweep Surface* (p. 1409).

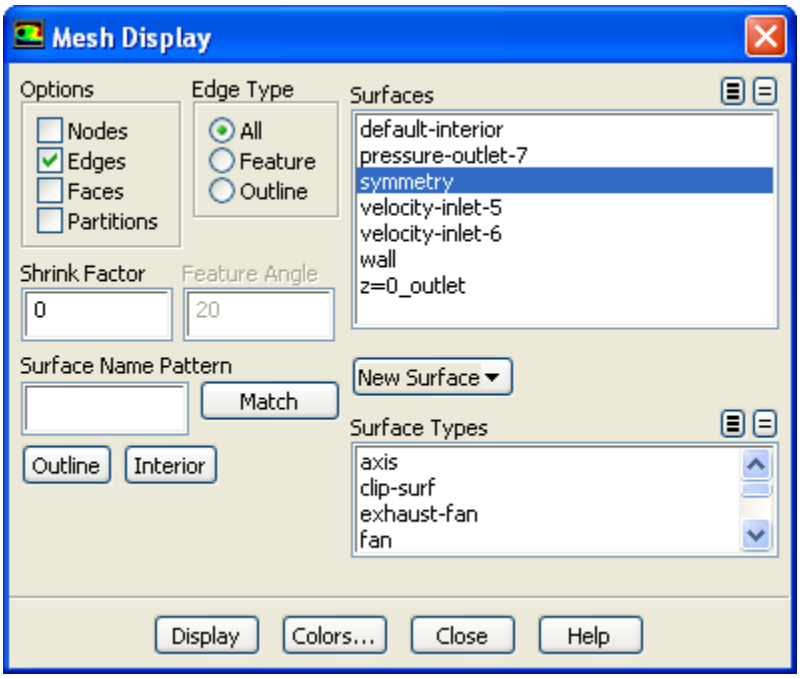
31.1.1.1. Generating Mesh or Outline Plots

You can draw the mesh or outline for all or part of your domain using the *Mesh Display Dialog Box* (p. 1635) (*Figure 31.5* (p. 1382)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General → Display...

Figure 31.5 The Mesh Display Dialog Box



The basic steps for generating a mesh or outline plot are as follows:

1. Choose the surfaces for which you want to display the mesh or outline in the **Surfaces** list.

   If you want to select several surfaces of the same type, select that type in the **Surface Types** list instead. All of the surfaces of that type will be selected automatically in the **Surfaces** list (or deselected, if they are all selected already).

   Another shortcut is to specify a **Surface Name Pattern** and click **Match** to select surfaces with names that match the specified pattern. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (e.g., **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).

   To choose all "outline" surfaces (i.e., surfaces on the outer boundary of the domain), click the **Outline** button below the **Surface Types** list. If all outline surfaces are already selected, this will deselect them. To choose all "interior" surfaces, click the **Interior** button. If all interior surfaces are already selected, this will deselect them.

2. Depending on what you want to draw, do one or more of the following:

   - To draw an outline of the selected surfaces (as in *Figure 31.1* (p. 1380)), select **Edges** under **Options** and **Outline** under **Edge Type**. If you need more detail in the outline display of a complex geometry, see the description of the **Feature** option, below.
   - To draw the mesh edges (as in *Figure 31.2* (p. 1380)), select **Edges** under **Options** and **All** under **Edge Type**.
   - To generate a filled-mesh display (as in *Figure 31.3* (p. 1381)), select **Faces** under **Options**.
   - To draw the nodes on the selected surfaces (as in *Figure 31.4* (p. 1381)), select **Nodes** under **Options**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set any of the mesh and outline display options described in the following section.
4. Click the **Display** button to draw the specified mesh or outline in the active graphics window.

To display filled meshes, with smoothly shaded display, enable lighting and select a lighting interpolation method other than **Flat** in the *Display Options Dialog Box* (p. 1992) or the *Lights Dialog Box* (p. 2005).

If you display nodes, and you want to change the symbol representing the nodes, you can change the **Point Symbol** in the *Display Options Dialog Box* (p. 1992). See *Modifying the Rendering Options* (p. 1425) for details.

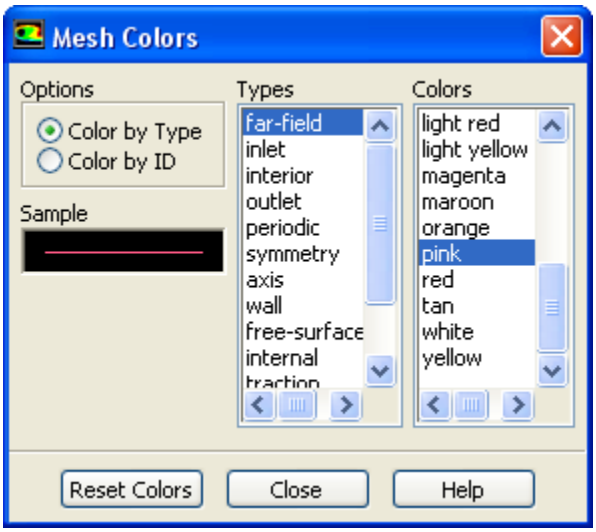
31.1.1.2. Mesh and Outline Display Options

The options mentioned in the procedure in the previous section include modifying the mesh colors, adding the outline of important features to an outline display, drawing partition boundaries, and shrinking the faces and/or cells in the display.

31.1.1.2.1. Modifying the Mesh Colors

ANSYS FLUENT allows you to control the colors that are used to render the meshes for each zone type or surface. This capability can help you to understand mesh plots quickly and easily. To modify the colors, open the *Mesh Colors Dialog Box* (p. 1638) (*Figure 31.6* (p. 1383)) by clicking on the **Colors...** button in the *Mesh Display Dialog Box* (p. 1635).

Figure 31.6 The Mesh Colors Dialog Box



You can set colors individually for the meshes displayed on each surface, using the *Scene Description Dialog Box* (p. 1994).

By default, the **Color by Type** option is turned on, allowing you to assign colors based on zone type. To change the color used to draw the mesh for a particular zone type, select the zone type in the **Types** list and then select the new color in the **Colors** list. You will see the effect of your change when you next display the mesh. Note that the **surface** type in the **Types** list applies to all surface meshes (i.e., meshes that are drawn for surfaces created using the dialog boxes opened from the **Surface** menu) except zone surfaces.

If you prefer to use the colors ANSYS FLUENT assigns by zone ID, then you can display the mesh using the **Color by ID** option.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.1.2.2. Adding Features to an Outline Display

For closed 3D geometries such as cylinders, the standard outline display often will not show enough detail to accurately depict the shape. This is because for each boundary, only those edges on the "outside" of the geometry (i.e., those that are used by only one face on the boundary) are drawn. In *Figure 31.7* (p. 1384), which shows the outline display for a complicated duct geometry, only the inlet and outlet are visible.

Figure 31.7 Standard Outline of Complex Duct

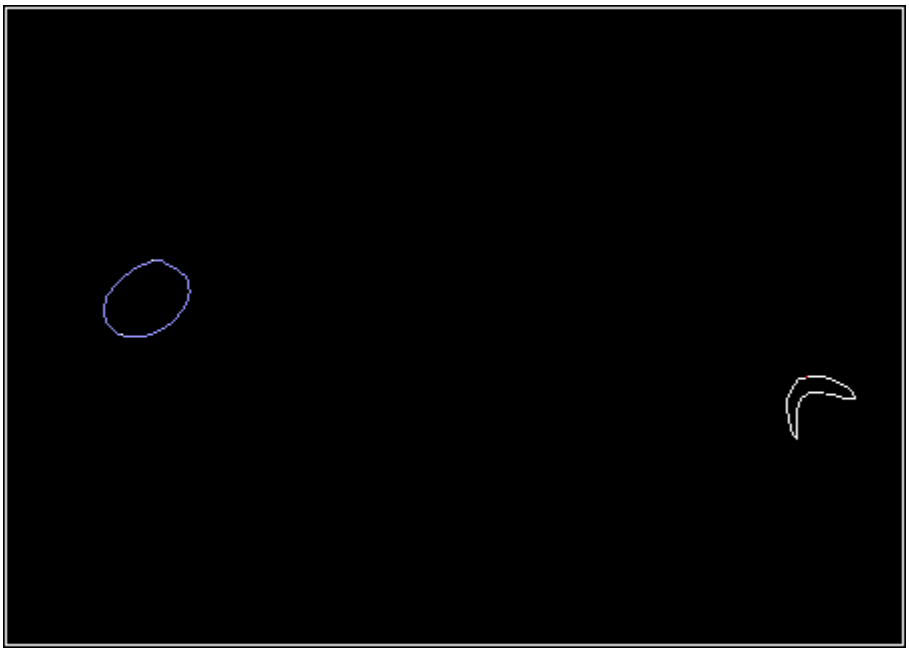
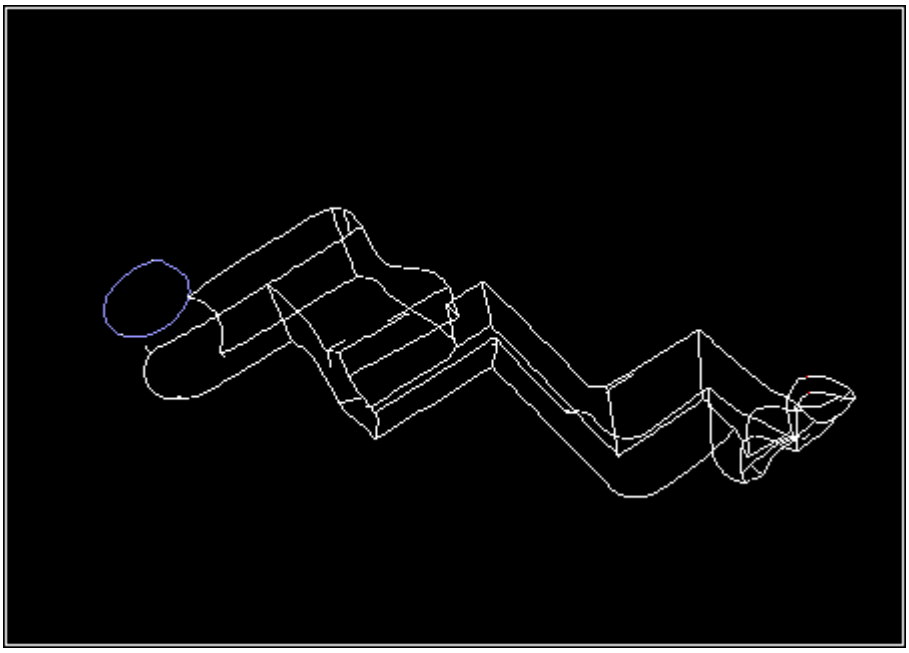


Figure 31.8 Feature Outline of Complex Duct



You can capture additional features using the **Feature** option in the *Mesh Display Dialog Box* (p. 1635). (See *Figure 31.8* (p. 1384).) Enable **Feature** under **Edge Type**, and then set the **Feature Angle**. With the default

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature Angle of 20, if the difference between the normal directions of two adjacent faces is more than 20°, the edge between those faces will be drawn. Decreasing the **Feature Angle** will result in more edge lines (i.e., more detail) being added to the outline display. The appropriate angle for your geometry will depend on its curvature and complexity. You can modify the **Feature Angle** until you find the value that yields the best outline display.

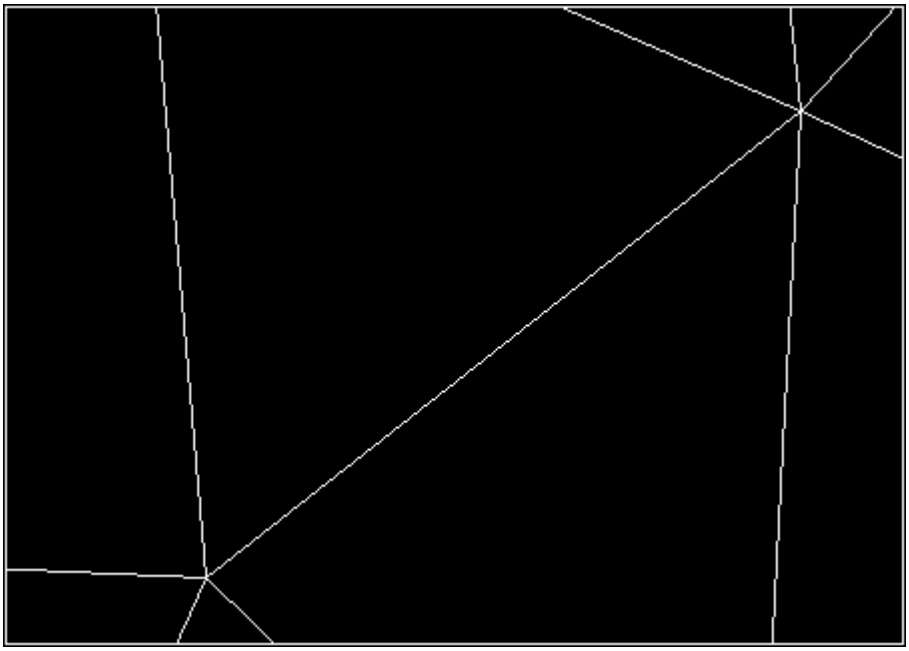
31.1.1.2.3. Drawing Partition Boundaries

If you have partitioned your mesh for parallel processing, you can add the display of partition boundaries to the mesh display by turning on the **Partitions** option in the *Mesh Display Dialog Box* (p. 1635).

31.1.1.2.4. Shrinking Faces and Cells in the Display

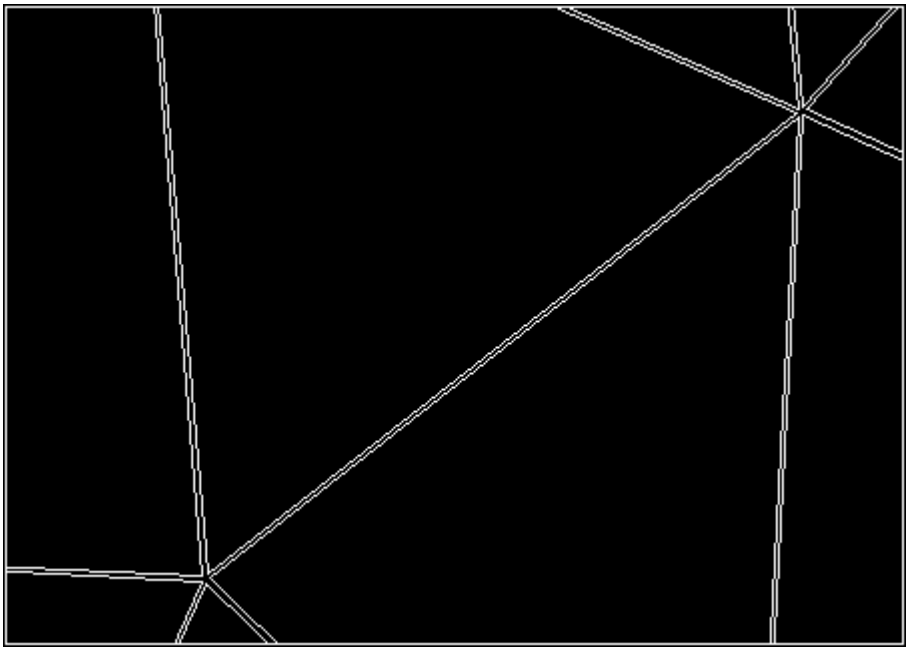
To distinguish individual faces or cells in the display, enlarge the space between adjacent faces or cells by increasing the **Shrink Factor** in the *Mesh Display Dialog Box* (p. 1635). The default value of zero produces a display in which the edges of adjacent faces or cells overlap. A value of 1 creates the opposite extreme: each face or cell is represented by a point and there is considerable space between each one. A small value such as 0.01 may be large enough to allow you to distinguish one face or cell from its neighbor. Displays with different **Shrink Factor** values are shown in *Figure 31.9* (p. 1385) and *Figure 31.10* (p. 1386). Click **Display** to see the effect of the change in **Shrink Factor**.

Figure 31.9 Mesh Display with Shrink Factor = 0



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.10 Mesh Display with Shrink Factor = 0.01



31.1.2. Displaying Contours and Profiles

ANSYS FLUENT allows you to plot contour lines or profiles superimposed on the physical domain. Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, etc.). A profile plot draws these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface. Sample plots are shown in *Figure 31.11* (p. 1387) and *Figure 31.12* (p. 1387).

For information about displaying contours or profiles on a surface that sweeps through the domain, see *Displaying Results on a Sweep Surface* (p. 1409).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.11 Contours of Static Pressure

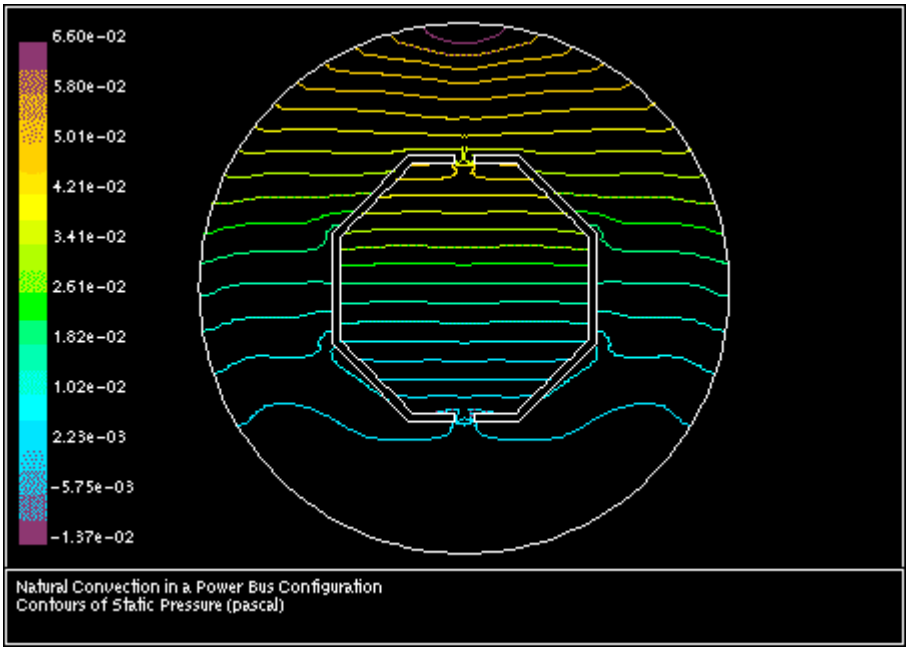
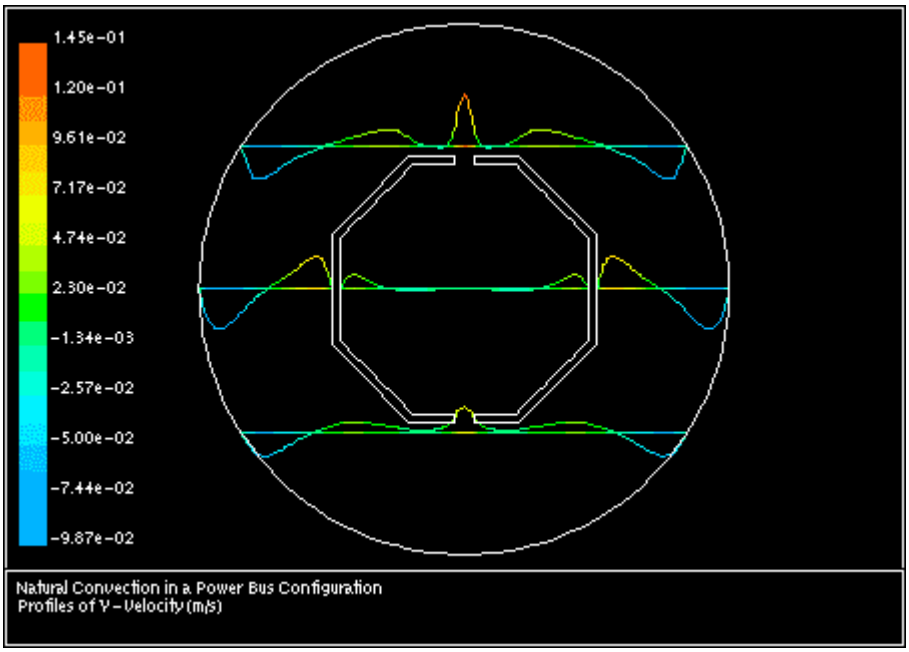


Figure 31.12 Profile Plot of y Velocity



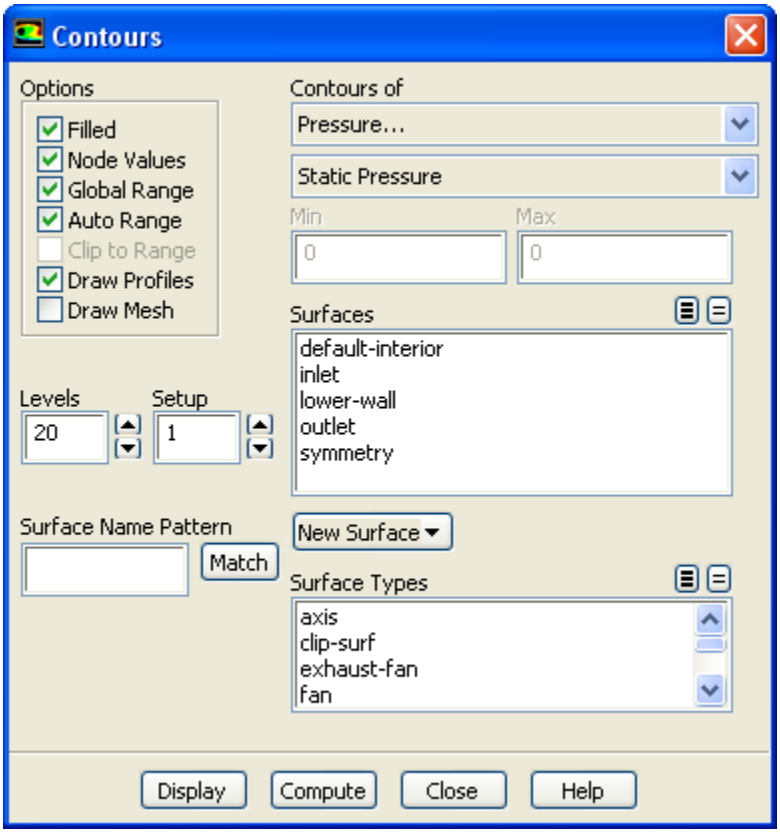
31.1.2.1. Generating Contour and Profile Plots

You can plot contours or profiles using the *Contours Dialog Box* (p. 1967) (*Figure 31.13* (p. 1388)).

Graphics and Animations → **Contours** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.13 The Contours Dialog Box



The basic steps for generating a contour or profile plot are as follows:

1. Select the variable or function to be contoured or profiled in the **Contours of** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)

2. Choose the surface or surfaces on which to draw the contours or profiles in the **Surfaces** list. For 2D cases, if no surface is selected, contouring or profiling is done on the entire domain. For 3D cases, you must always select at least one surface.

   If you want to select several surfaces of the same type, you can select that type in the **Surface Types** list instead. All of the surfaces of that type will be selected automatically in the **Surfaces** list (or deselected, if they are all selected already).

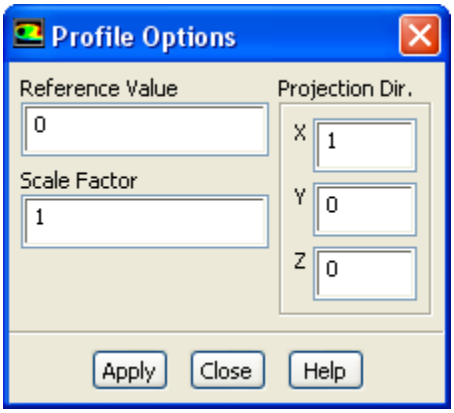
   Another shortcut is to specify a **Surface Name Pattern** and click **Match** to select surfaces with names that match the specified pattern. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (e.g., **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).

3. Specify the number of contours or profiles in the **Levels** field. The maximum number of levels allowed is 100.

4. If you are generating a profile plot, enable the **Draw Profiles** option. In the resulting *Profile Options Dialog Box* (p. 1970) (*Figure 31.14* (p. 1389)) you will define the profiles.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.14 The Profile Options Dialog Box



a. Set the "zero height" reference value for the profile (**Reference Value**) and the length scale factor for projection (**Scale Factor**). Any point on the profile with a value equal to the **Reference Value** will be plotted exactly on the defining surface. Values greater than the **Reference Value** will be projected ahead of the surface (in the direction of **Projection Dir.**) and scaled by **Scale Factor**), and values less than the **Reference Value** will be projected behind the surface and scaled.

   These parameters can be used to create fuller profiles when you need to display the variation in a variable which is small compared to the absolute value of the variable. Consider, for example, the display of temperature profiles when the temperature range in the domain is from 300 K to 310 K. The 10 K range in the temperature will be hard to detect when profiles are drawn using the default scaling (which will be based on the absolute magnitude of 310 K). To create a fuller profile, you can set the **Reference Value** to 300 and the profile **Scaling Factor** to 5 (for example) to magnify the display of the remaining 10 K range. In subsequent display of the profiles, the reference value of 300 will be effectively subtracted from the data before display so that the temperatures of 300 K will not be offset from the baselines. The profiles will then reflect only the variation of temperature from 300 K.

b. Set the direction in which profiles are projected (**Projection Dir.**). In 2D, for example, a contour plot of pressure on the entire domain can be projected in the z direction to form a carpet plot, or a contour plot of y velocity on a sequence of y-coordinate slice lines can be projected in the y direction to form a series of velocity profiles (as shown in *Figure 31.12* (p. 1387)).

c. Click **Apply** and close the *Profile Options Dialog Box* (p. 1970).

5. Set any of the contour and profile plot options described below.

6. Click the **Display** button to draw the specified contours or profiles in the active graphics window.

The resulting display will include the specified number of contours or profiles of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

31.1.2.2. Contour and Profile Plot Options

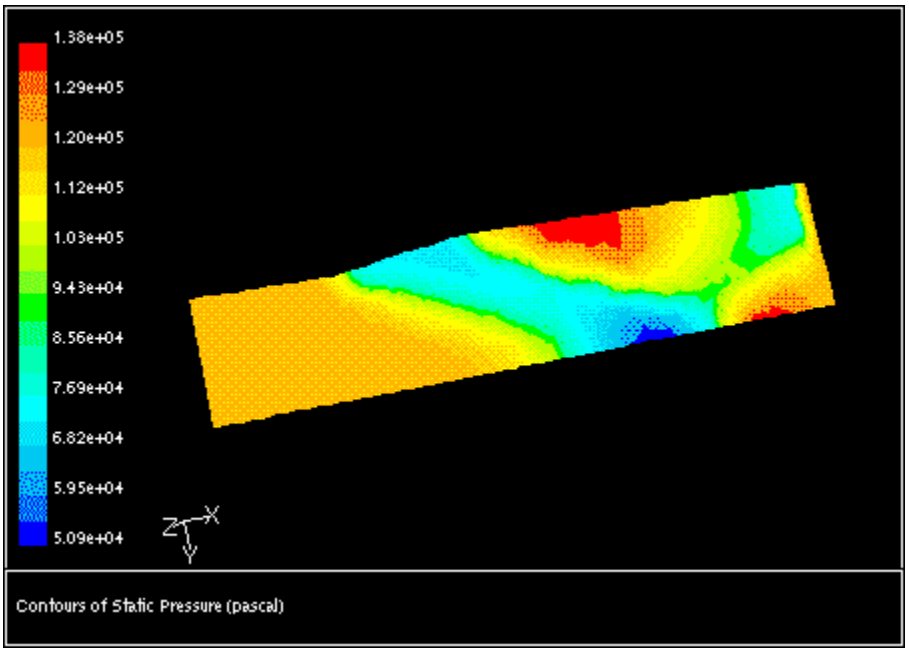
The options mentioned in the procedure above include drawing color-filled contours/profiles (instead of the default line contours/profiles), specifying a range of values to be contoured or profiled, including portions of the mesh in the contour or profile display, choosing node or cell values for display, and storing the contour or profile plot settings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.2.2.1. Drawing Filled Contours or Profiles

Color-filled contour or profile plots show a contour or profile display containing a continuous color display (see *Figure 31.15* (p. 1390)), instead of just drawing lines representing specific values. Note that a color-filled profile display is often referred to as a "carpet plot". To generate a filled contour or profile plot, enable the **Filled** option in the *Contours Dialog Box* (p. 1967) during step 5 in the previous section.

Figure 31.15 Filled Contours of Static Pressure



To display smoothly shaded filled contours, you must enable lighting and select a lighting interpolation method other than **Flat** in the *Display Options Dialog Box* (p. 1992) or the *Lights Dialog Box* (p. 2005). You will not get smooth shading of filled contours if the **Clip to Range** option is turned on. Smooth shading of filled profiles is not available.

31.1.2.2.2. Specifying the Range of Magnitudes Displayed

By default, the minimum and maximum values contoured or profiled are set based on the range of values in the entire domain. This means that the color scale will start at the smallest value in the domain (shown in the **Min** field) and end at the largest value (shown in the **Max** field). If you are plotting contours or profiles on a subset of the domain (i.e., on a surface), your plot may cover only the midrange of the color scale. For example, if blue corresponds to 0 and red corresponds to 10, and the values on your surface range only from 4 to 6, your plot will contain mostly green contours or profiles, since green is the color at the middle of the default color scale.

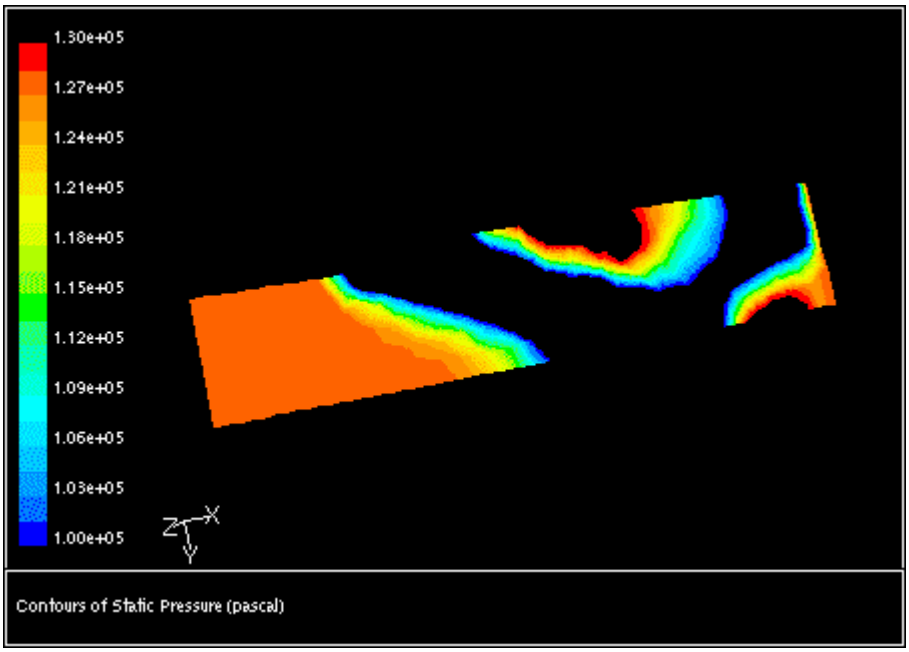
- To focus in on a smaller range of values, so that blue corresponds to 4 and red to 6, manually reset the range to be displayed. You can also use the minimum and maximum values on the selected surfaces—rather than in the entire domain—to determine the range. Another reason to manually set the range is if you are interested only in certain values. For example, if you want to determine the region where pressure exceeds a certain value, you can increase the minimum value for display so that the lower pressure values are not displayed.
- To manually set the contour/profile range, turn off the **Auto Range** option in the *Contours Dialog Box* (p. 1967). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be displayed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - To show the default range at any time, click the **Compute** button and the **Min** and **Max** fields will be updated.
  - If you are drawing filled contours or profiles you can control whether or not values outside the prescribed **Min**/ **Max** range are displayed.

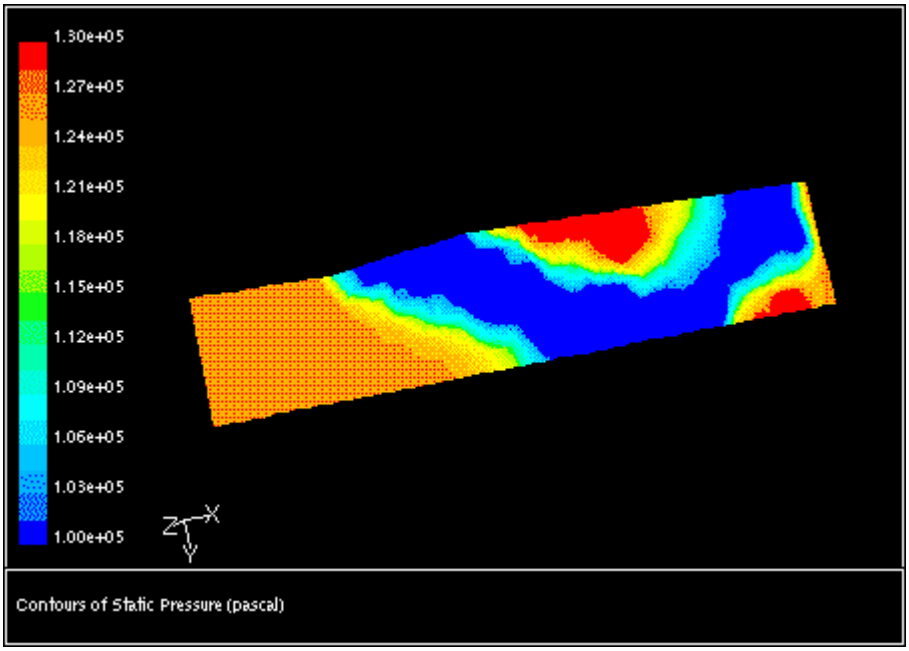
- To leave areas in which the value is outside the specified range empty (i.e., draw no contours or profiles), enable the **Clip to Range** option. This is the default setting. If you turn **Clip to Range** off, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. *Figure 31.16* (p. 1391) and *Figure 31.17* (p. 1392) show the results of enabling/disabling the **Clip to Range** option.
- To base the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain, turn off the **Global Range** option in the *Contours Dialog Box* (p. 1967). The **Min** and **Max** values will be updated when you next click **Compute** or **Display**.

Figure 31.16 Filled Contours with Clip to Range On



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.17 Filled Contours with Clip to Range Off



31.1.2.2.3. Including the Mesh in the Contour Plot

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your contour or profile plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the contours. This is accomplished by turning on the **Draw Mesh** option in the *Contours Dialog Box* (p. 1967). The *Mesh Display Dialog Box* (p. 1635) will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the *Contours Dialog Box* (p. 1967), the mesh display, as defined in the *Mesh Display Dialog Box* (p. 1635), will be included in the contour or profile plot.

31.1.2.2.4. Choosing Node or Cell Values

In ANSYS FLUENT you can choose to display the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are displayed. For line contours or profiles, node values are always used. To display filled contours or profiles and to display the cell values, turn off the **Node Values** option. Filled contours/profiles of node values will show a smooth gradation of color, while filled contours/profiles of cell values may show sharp changes in color from one cell to the next.

For face-only functions (e.g., **Wall Shear Stress**), the cell values that are displayed for boundary zone surfaces will actually be the face values. This is only true in the case of boundary zone surfaces created for postprocessing, where the actual cell values are used for the part of the surface which lies in the interior. These face values are more accurate, as face-only functions are computed on the faces and not on the cells. For more information about cell values, see *Cell Values* (p. 1529).

If you are plotting contours to show the effect of a porous medium or fan, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.2.2.5. Storing Contour Plot Settings

For frequently used combinations of contour variables and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the *Contours Dialog Box* (p. 1967). When you click on the **Display** button, the settings for **Options**, **Contours of**, **Min**, **Max**, and **Surfaces** will be saved. You can then change the **Setup** number to an unused value (i.e., an ID for which no information has been saved) and generate a different contour plot. To generate a plot using the saved setup information, change the **Setup** number back to the value for which you saved contour information and click **Display**. You can save up to 10 different setups.

Important

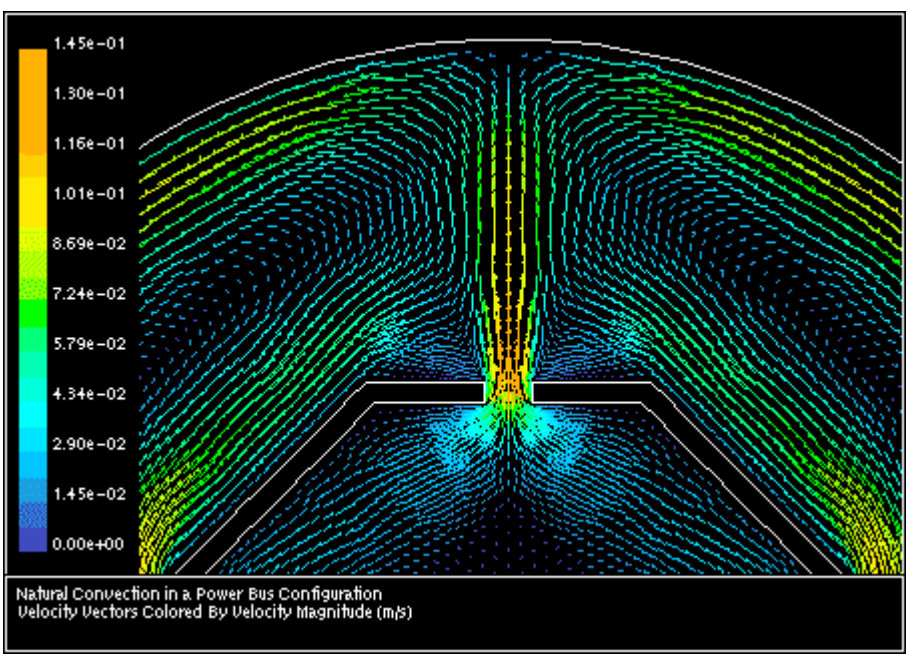
Note that the number of contour **Levels**, the surfaces selected for display in the *Mesh Display Dialog Box* (p. 1635) (when the **Draw Mesh** option is activated), and the settings for profiles in the *Profile Options Dialog Box* (p. 1970) (when the **Draw Profiles** option is activated) will *not* be saved in the **Setup**, nor will the **Setup** be saved in the case file.

31.1.3. Displaying Vectors

You can draw vectors in the entire domain, or on selected surfaces. By default, one vector is drawn at the center of each cell (or at the center of each facet of a data surface), with the length and color of the arrows representing the velocity magnitude (*Figure 31.18* (p. 1393)). The spacing, size, and coloring of the arrows can be modified, along with several other vector plot settings. Velocity vectors are the default, but you can also plot vector quantities other than velocity. Note that cell-center values are always used for vector plots; you cannot plot node-averaged values.

For information about displaying vectors on a surface that sweeps through the domain, see *Displaying Results on a Sweep Surface* (p. 1409).

Figure 31.18 Velocity Vector Plot



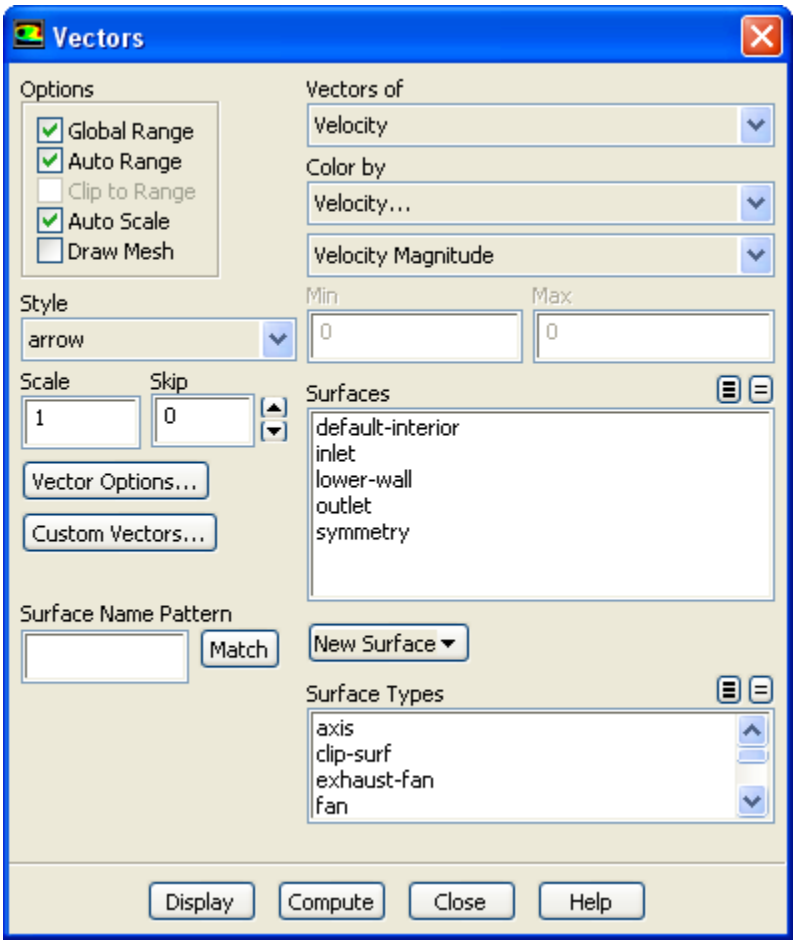
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.3.1. Generating Vector Plots

You can plot vectors using the *Vectors Dialog Box* (p. 1970) (*Figure 31.19* (p. 1394)).

Graphics and Animations → **Vectors** → **Set Up...**

Figure 31.19 The Vectors Dialog Box



The procedure for generating a vector plot is as follows:

1. In the **Vectors of** drop-down list, select the vector quantity to be plotted. By default, only velocity and relative velocity are available, but you can create your own custom vectors as described in *Creating and Managing Custom Vectors* (p. 1398).
2. In the **Surfaces** list, choose the surface(s) on which you want to display vectors. If you want to display vectors on the entire domain, select none of the surfaces in the list.

   If you want to select several surfaces of the same type, you can select that type in the **Surface Types** list instead. All of the surfaces of that type will be selected automatically in the **Surfaces** list (or deselected, if they are all selected already).

   Another short cut is to specify a **Surface Name Pattern** and click **Match** to select surfaces with names that match the specified pattern. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (e.g., **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set any of the vector plot options as described in *Vector Plot Options* (p. 1395).
4. Click the **Display** button to draw the vectors in the active graphics window.

31.1.3.2. Displaying Relative Velocity Vectors

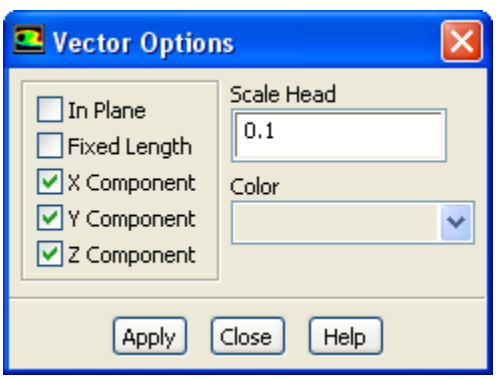
If you are solving your problem using one or more moving reference frames or moving meshes, you will have the option to display either the absolute vectors or the relative vectors. If you select **Velocity** (the default) in the **Vectors of** list, the vectors will be drawn based on the absolute, stationary reference frame. If you select **Relative Velocity**, the vectors will be drawn based on the reference frame of the **Reference Zone** in the *Reference Values Task Page* (p. 1898). See *Setting the Reference Zone* (p. 1525) for details. (If you are modeling a single moving reference frame, you need not specify the **Reference Zone**; the vectors will be drawn based on the moving reference frame.)

31.1.3.3. Vector Plot Options

The options mentioned in the procedure above include scaling the vector arrows, skipping the display of some vectors, displaying vectors in the plane of the data surface, displaying fixed-length or fixed-color vectors, displaying directional components of the vectors, specifying a range of values to be displayed, coloring the vectors by a different scalar field, including portions of the mesh in the vector display, and changing the style of the arrows or the scale of the arrowheads.

The most common options are set in the *Vectors Dialog Box* (p. 1970), and others are set in the *Vector Options Dialog Box* (p. 1973) (*Figure 31.20* (p. 1395)), which you can open by clicking on the **Vector Options...** button in the *Vectors Dialog Box* (p. 1970).

Figure 31.20 The Vector Options Dialog Box



31.1.3.3.1. Scaling the Vectors

By default, vectors are scaled automatically so that the arrows overlap minimally when no vectors are skipped. For instructions on thinning the vector display, see *"Thinning" Pathlines* (p. 1404). With the **Auto Scale** option, you can modify the **Scale** factor (which is set to 1 by default) to increase or decrease the vector scale from the default "auto scale". The main advantage of autoscaling is that the vector display with a scale factor of 1 will always be appropriate, regardless of the size of the domain, giving you a better starting point for fine-tuning the vector scale.

If you turn off the **Auto Scale** option, the vectors will be drawn at their actual sizes scaled by the scale factor (**Scale**, which is set to 1 by default). The "actual" size of a vector is the magnitude of the vector variable (velocity, by default) at the point where it is drawn. A vector drawn at a point where the velocity magnitude is 100 m/s is drawn 100 m long, whether the domain is 0.1 m or 1000 m. You can modify the vector scale

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by changing the value of **Scale** in the *Vectors Dialog Box* (p. 1970) until the size of the vectors (i.e., the actual size multiplied by **Scale**) is satisfactory.

31.1.3.3.2. Skipping Vectors

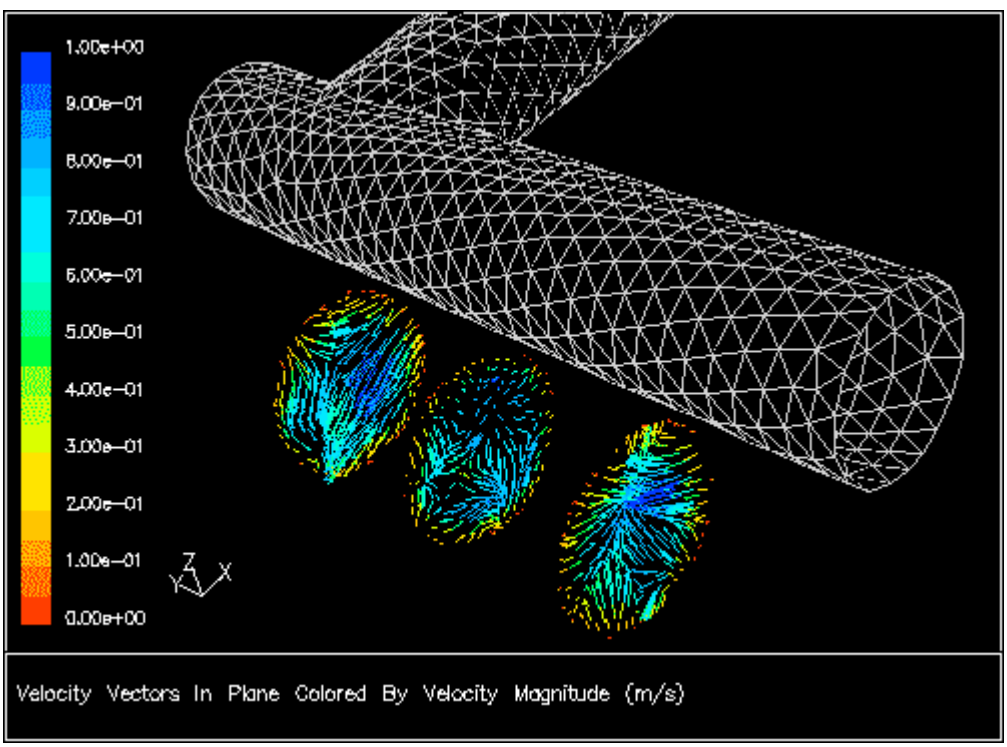
If your vector display is difficult to understand because there are too many arrows displayed, you can "thin out" the vectors by changing the **Skip** value in the *Vectors Dialog Box* (p. 1970). By default, **Skip** is set to 0, indicating that a vector will be drawn for each cell in the domain or for each face on the selected surface (e.g., n vectors). If you increase **Skip** to 1, every other vector will be displayed, yielding $n/2$ vectors. If you increase **Skip** to 2, every third vector will be displayed, yielding $n/3$ vectors, and so on. The order of faces on the selected surface (or cells in the domain) will determine which vectors are skipped or drawn; thus adaption and reordering will change the appearance of the vector display when a non-zero **Skip** value is used.

31.1.3.3.3. Drawing Vectors in the Plane of the Surface

For some problems, you may be interested in visualizing velocity (or other vector) components that are normal to the flow. These "secondary flow" components are usually much smaller than the components in the flow direction and are difficult to see when the flow direction components are also visible. To easily view the normal flow components, you can enable the **In Plane** option in the *Vector Options Dialog Box* (p. 1973). When this option is on, ANSYS FLUENT will display only the vector components in the plane of the surface selected for display. If the selected surface is a cross-section of the flow domain, you will be displaying the components normal to the flow.

Figure 31.21 (p. 1396) shows velocity vectors generated using the **In Plane** option. Note that these vectors have been translated outside the domain, as described in *Transforming Geometric Objects in a Scene* (p. 1448), so that they can be seen more easily.

Figure 31.21 Velocity Vectors Generated Using the In Plane Option



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.3.3.4. Displaying Fixed-Length Vectors

By default, the length of a vector is proportional to its velocity magnitude. If you want all of the vectors to be displayed with the same length, you can enable the **Fixed Length** option in the *Vector Options Dialog Box* (p. 1973). To modify the vector length, adjust the value of the **Scale** factor in the *Vectors Dialog Box* (p. 1970).

31.1.3.3.5. Displaying Vector Components

All Cartesian components of the vectors are drawn by default, so that the arrow points along the resultant vector in physical space. However, sometimes one of the components, say, the x component, is relatively large. In such cases, you may want to suppress the x component and scale up the vectors, in order to visualize the smaller y and z components. To suppress one or more of the vector components, turn off the appropriate button(s) (**X**, **Y**, or **Z Component**) in the *Vector Options Dialog Box* (p. 1973).

31.1.3.3.6. Specifying the Range of Magnitudes Displayed

By default, the minimum and maximum vectors included in the vector display are set based on the range of vector-variable (velocity, by default) magnitudes in the entire domain. If you want to focus in on a smaller range of values, you can restrict the range to be displayed. The color scale for the vector display will change to reflect the new range of values. (You can also use the minimum and maximum values on the selected surfaces—rather than on the entire domain—to determine the range, or change the scalar field by which the vectors are colored from velocity magnitude to any other scalar, as described below.)

To manually set the range of velocity magnitudes (or the range of whatever scalar field is selected in the **Color by** drop-down list), turn off the **Auto Range** option in the *Vectors Dialog Box* (p. 1970). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be displayed. For example, if you want to display velocity vectors only in regions where the velocity magnitude exceeds 150 m/s but is less than 300 m/s, you will change the value of **Min** to 150 and the value of **Max** to 300. Similarly, if you are coloring the vectors by static pressure, you can choose to display velocity vectors only in regions where the pressure is within a specified range. To show the default range at any time, click the **Compute** button and the **Min** and **Max** fields will be updated.

When you restrict the range of vectors displayed, you can also control whether or not values outside the prescribed **Min**/ **Max** range are displayed. To leave areas in which the value is outside the specified range empty (i.e., draw no vectors), enable the **Clip to Range** option. This is the default setting. If you turn **Clip to Range** off, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. This feature is the same as the one available for displaying filled contours (see *Figure 31.16* (p. 1391) and *Figure 31.17* (p. 1392)).

You can also choose to base the minimum and maximum values on the range of values on the selected surfaces, rather than the entire domain. To do this, turn off the **Global Range** option in the *Vectors Dialog Box* (p. 1970). The **Min** and **Max** values will be updated when you next click **Compute** or **Display**.

31.1.3.3.7. Changing the Scalar Field Used for Coloring the Vectors

If you want to color the vectors by a scalar field other than velocity magnitude (the default), you can select a different variable or function in the **Color by** drop-down list. Select the desired category in the upper list, and then choose one of the related quantities from the lower list. If you choose static pressure, for example, the length of the vectors will still correspond to the velocity magnitude, but the color of the vectors will correspond to the value of pressure at each point where a vector is drawn.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.3.3.8. Displaying Vectors Using a Single Color

If you want all of the vectors to be the same color, you can select the color to be used in the **Color** drop-down list in the *Vector Options Dialog Box* (p. 1973). If no color is selected (i.e., if you choose the empty space at the top of the drop-down list—the default selection), the vector color will be determined by the **Color by** field specified in the *Vectors Dialog Box* (p. 1970). Single color vectors are useful in displays that overlay contours and vectors.

31.1.3.3.9. Including the Mesh in the Vector Plot

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your vector plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the vectors. This is accomplished by turning on the **Draw Mesh** option in the *Vectors Dialog Box* (p. 1970). The *Mesh Display Dialog Box* (p. 1635) will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the *Vectors Dialog Box* (p. 1970), the mesh display, as defined in the *Mesh Display Dialog Box* (p. 1635), will be included in the vector plot.

31.1.3.3.10. Changing the Arrow Characteristics

There are five different styles available for drawing the vector arrows. Choose **cone**, **filled-arrow**, **arrow**, **harpoon**, or **headless** in the **Style** drop-down list in the *Vectors Dialog Box* (p. 1970). The default arrow style is **harpoon**.

If you choose a vector arrow style that includes heads, you can control the size of the arrowhead by modifying the **Scale Head** value in the *Vector Options Dialog Box* (p. 1973).

31.1.3.4. Creating and Managing Custom Vectors

In addition to the velocity vector quantity provided by ANSYS FLUENT, you can also define your own custom vectors to be plotted. This capability is available with the *Custom Vectors Dialog Box* (p. 1973).

Any custom vectors that you define will be saved in the case file the next time that you save it. You can also save your custom vectors to a separate file, so that they can be used with a different case file.

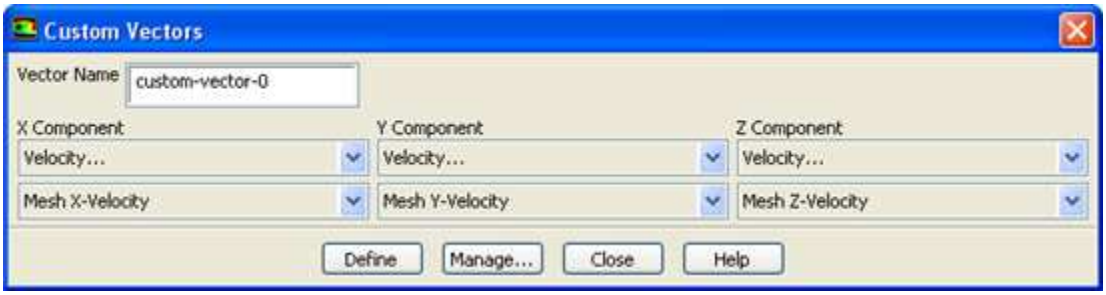
31.1.3.4.1. Creating Custom Vectors

To create your own custom vector, you will use the *Custom Vectors Dialog Box* (p. 1973) (*Figure 31.22* (p. 1399)). This dialog box allows you to define custom vectors based on existing quantities. Any vectors that you define will be added to the **Vectors of** list in the *Vectors Dialog Box* (p. 1970).

To open the *Custom Vectors Dialog Box* (p. 1973), click the **Custom Vectors...** button in the *Vectors Dialog Box* (p. 1970).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.22 The Custom Vectors Dialog Box



The steps for creating a custom vector are as follows:

1. Specify the name of the custom vector in the **Vector Name** field.

   Important

   Do not specify a name that is already used for a standard vector (e.g., `velocity` or `relative-velocity`).

2. Select the variable or function for the x component of the vector in the **X Component** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. For an explanation of the variables in the list, see *Field Function Definitions* (p. 1529).
3. Repeat the step above to select the variable or function for the y component (and, in 3D, the z component) of the custom vector.

   Important

   You can use the **Custom Vectors** option to plot vectors in solid cell zones. The scalars that are selected in the x, y components (and, in 3D, the z component), and which are valid in solid regions, will have vector plots displayed in the solid cell zones. Note that if a vector has no valid components in the solid region, then that vector will not be plotted in the solid region. However, if at least one component of the vector is valid in the solid region, then only that component of the vector will be plotted.

4. Click the **Define** button.

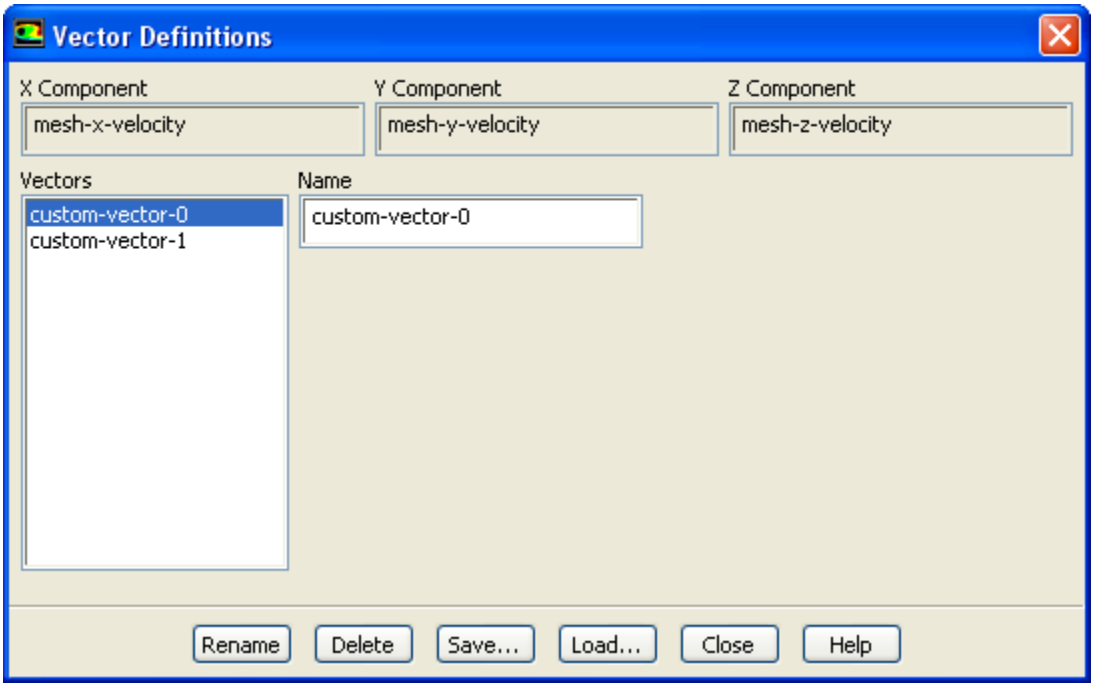
31.1.3.4.2. Manipulating, Saving, and Loading Custom Vectors

Once you have defined your vectors, you can manipulate them using the *Vector Definitions Dialog Box* (p. 1974) (*Figure 31.23* (p. 1400)). You can display a vector definition to be sure that it is correct, delete the vector if you decide that it is incorrect and needs to be redefined, or give the vector a new name. You can also save custom vectors to a file or read them from a file. The custom vector file allows you to transfer custom vectors between case files.

To open the *Vector Definitions Dialog Box* (p. 1974), click the **Manage...** button in the *Custom Vectors Dialog Box* (p. 1973).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.23 The Vector Definitions Dialog Box



The following actions can be performed in the *Vector Definitions Dialog Box* (p. 1974):

- To check the definition of a vector, select it in the **Vectors** list. Its definition will be displayed in the **X Component**, **Y Component**, and **Z Component** fields at the top of the dialog box. This display is for informational purposes only; you cannot edit it. If you want to change a vector definition, you must delete the vector and define it again in the *Custom Vectors Dialog Box* (p. 1973).
- To delete a vector, select it in the **Vectors** list and click the **Delete** button.
- To rename a vector, select it in the **Vectors** list, enter a new name in the **Name** field, and click the **Rename** button.

Important

Do not specify a name that is already used for a standard vector (e.g., `velocity` or `relative-velocity`).

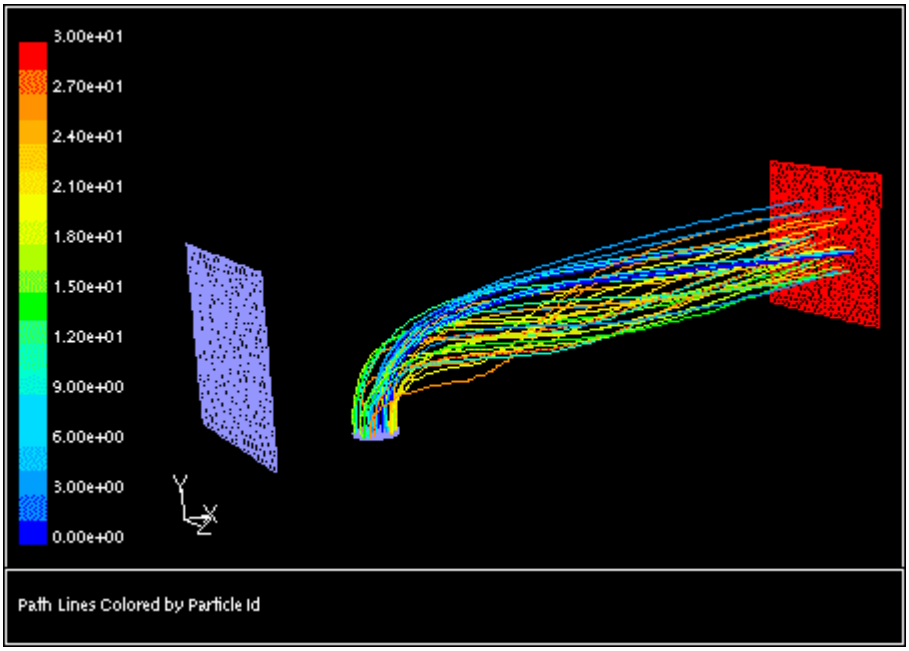
- To save all the vectors in the **Vectors** list to a file, click the **Save...** button, and specify the file name in *The **Select File** Dialog Box* (p. 33).
- To read custom vectors from a file that you saved as described above, click the **Load...** button and specify the file name in *The **Select File** Dialog Box* (p. 33). (Custom vectors are valid Scheme functions, and can also be loaded with the **File/Read/Scheme...** menu item, as described in *Reading Scheme Source Files* (p. 73).)

31.1.4. Displaying Pathlines

Pathlines are used to visualize the flow of massless particles in the problem domain. The particles are released from one or more surfaces that you have created with the tools in the **Surface** menu (see *Creating Surfaces for Displaying and Reporting Data* (p. 1355)). A **line** or **rake** surface (see *Line and Rake Surfaces* (p. 1361)) is most commonly used. *Figure 31.24* (p. 1401) shows a sample plot of pathlines.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.24 Pathline Plot



Note that the display of discrete-phase particle trajectories is discussed in *Displaying of Trajectories* (p. 1068).

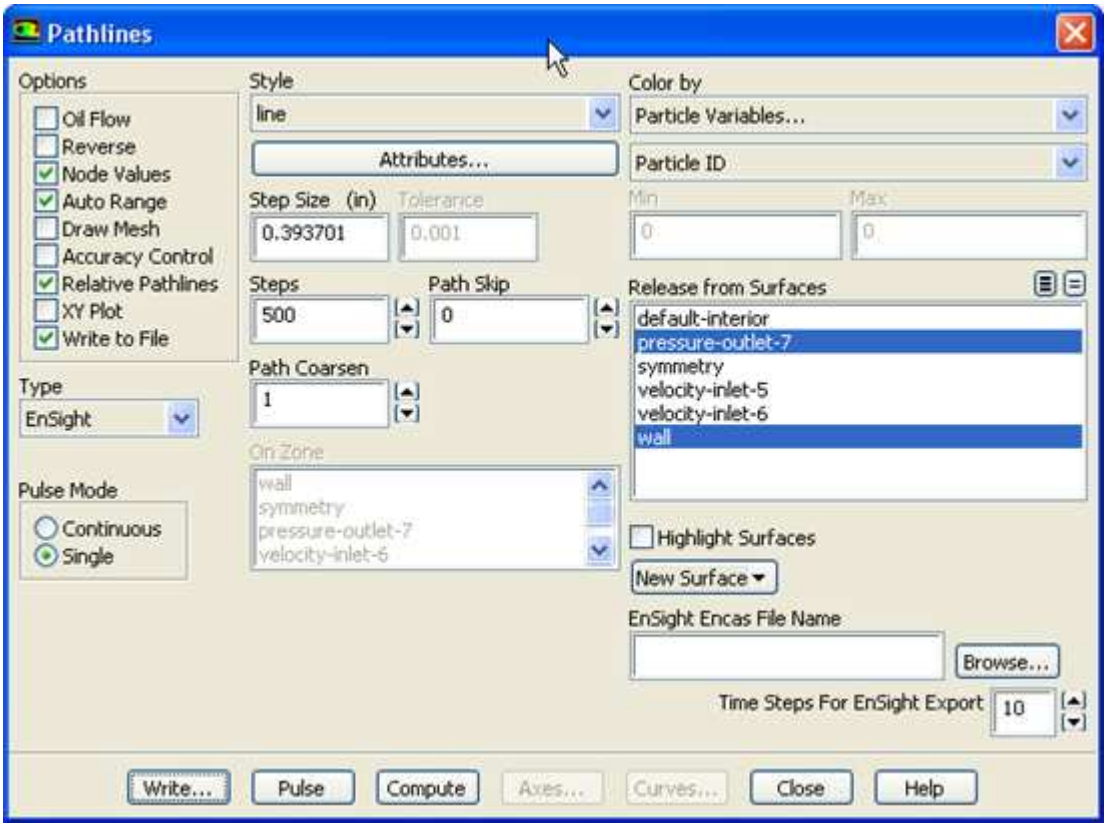
31.1.4.1. Steps for Generating Pathlines

You can plot pathlines using the *Pathlines Dialog Box* (p. 1975) (*Figure 31.25* (p. 1402)).

Graphics and Animations → **Pathlines** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.25 The Pathlines Dialog Box



The basic steps for generating pathlines are as follows:

1. Select the surface(s) from which to release the particles in the **Release From Surfaces** list.
2. Set the step size and the maximum number of steps. The **Step Size** sets the length interval used for computing the next position of a particle. Note that particle positions are always computed when particles enter/leave a cell; even if you specify a very large step size, the particle positions at the entry/exit of each cell will still be computed and displayed. The value of **Steps** sets the maximum number of steps a particle can advance. A particle will stop when it has traveled this number of steps or when it leaves the domain. One simple rule of thumb to follow when setting these two parameters is that if you want the particles to advance through a domain of length L, the **Step Size** times the number of **Steps** should be approximately equal to L.
3. Set any of the pathline plot options described in *Options for Pathline Plots* (p. 1402).
4. Click the **Display** button to draw the pathlines, or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

31.1.4.2. Options for Pathline Plots

You can include the mesh in the pathline display, control the style of the pathlines (including the twisting of ribbon-style pathlines), and color them by different scalar fields and control the color scale. You can also "thin" the pathline display, trace the particle positions in reverse, and draw "oil-flow" pathlines. If you are "pulsing" the pathlines, you can control the pulse mode. If you are using larger time step size for calculations then you can control the accuracy of the pathline by specifying tolerance. In addition to the regular pathline display, you can also generate an XY plot of a specified quantity along the pathline trajectories. Finally, you can choose node or cell values for display (or plotting).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.4.2.1. Including the Mesh in the Pathline Display

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your pathline display as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the pathlines (as in *Figure 31.24* (p. 1401)). This is accomplished by turning on the **Draw Mesh** option in the *Pathlines Dialog Box* (p. 1975). The *Mesh Display Dialog Box* (p. 1635) will appear when you enable the **Draw Mesh** option, where you can set the mesh display parameters. When you click **Display** in the *Pathlines Dialog Box* (p. 1975), the mesh display, as defined in the *Mesh Display Dialog Box* (p. 1635), will be included in the plot of pathlines.

31.1.4.2.2. Controlling the Pathline Style

Pathlines can be displayed as lines (with or without arrows), ribbons, cylinders (coarse, medium, or fine), triangles, spheres, or a set of points. You can choose **line**, **line-arrows**, **point**, **sphere**, **ribbon**, **triangle**, **coarse-cylinder**, **medium-cylinder**, or **fine-cylinder** in the **Style** drop-down list in the *Pathlines Dialog Box* (p. 1975). Pulsing can be done only on **point**, **sphere**, or **line** styles.

Once you have selected the pathline style, click the **Style Attributes...** button to set the pathline thickness and other parameters related to the selected **Style**:

- If you are using the **line** or **line-arrows** style, set the **Line Width** in the *Path Style Attributes Dialog Box* (p. 1979) that appears when you click the **Style Attributes...** button. For **line-arrows** you will also set the **Spacing Factor**, which controls the spacing between the lines. The size of the arrow heads can be adjusted by entering a value in the **Scale** text-entry box.
- If you are using the **point** style, you will set the **Marker Size** in the *Path Style Attributes Dialog Box* (p. 1979). The thickness of the pathline will be the thickness of the marker.
- If you are using the **sphere** style, you will set the **Diameter** and the **Detail** in the *Path Style Attributes Dialog Box* (p. 1979).

  The best diameter to use will depend on the dimensions of the domain, the view, and the particle density. However, an adequate starting point would be a diameter on the order of 1/4 of the average cell size or 1/4 step size. Units for the **Diameter** field correspond to the mesh dimensional units.

  The level of detail applied to the graphical rendering of the spheres can be controlled using the **Detail** field in the *Path Style Attributes Dialog Box* (p. 1979). The level of detail uses integer values ranging from 4 to 50. Note that the performance of the graphical rendering will be better when using a small level of detail, i.e., very coarse spheres, such as 6 or 8. The rendering performance significantly decreases with higher levels of detail. You should gradually increase the detail to determine the best-case scenario between performance and quality.

  Also note that to take full advantage of spherical rendering, lighting should be turned on in the view. The Gouraud setting provides much smoother looking spheres than the Flat setting and better performance than the Phong setting. For more information on lighting, see *Adding Lights* (p. 1423).
- If you are using the **triangle** or any of the **cylinder** styles, you will set the **Width** in the *Path Style Attributes Dialog Box* (p. 1979). For triangles, the specified value will be half the width of the triangle's base, and for cylinders, the value will be the cylinder's radius.
- If you are using the **ribbon** style, clicking on the **Style Attributes...** button will open the *Ribbon Attributes Dialog Box* (p. 1979), in which you can set the ribbon's **Width**. You can also specify parameters for twisting the ribbon pathlines. In the **Twist By** drop-down list, you can select a scalar field on which the pathline twisting is based (e.g., helicity). Select the desired category in the upper list and then select a related quantity in the lower list. The twisting may not be displayed smoothly because the scalar field by which you are twisting the pathline is calculated at cell centers only (and not interpolated to a particle's posi-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion). The **Twist Scale** sets the amount of twist for the selected scalar field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

When you click **Compute**, the **Min** and **Max** fields will be updated to show the range of the **Twist By** scalar field.

31.1.4.2.3. Controlling Pathline Colors

By default, the pathlines are colored by the particle ID number. That is, each particle's path will be a different color. You can also choose the color based on the surface from where the pathlines were released from using the surface ID as the particle variable. You can choose to color the pathlines by any of the scalar fields in the **Color by** drop-down list. Select the desired category in the upper list and then select a related quantity in the lower list. If you color the pathlines by velocity magnitude, for example, each particle's path will be colored depending on the speed of the particle at each point in the path.

The range of values of the selected scalar field will, by default, be the upper and lower limits of that field in the entire domain. The color scale will map to these values accordingly. If you prefer to restrict the range of the scalar field, turn off the **Auto Range** option (under **Options**) and set the **Min** and **Max** values manually beneath the **Color by** list. If you color the pathlines by velocity, and you limit the range to values between 30 and 60 m/s, for example, the "lowest" color will be used when the particle speed falls below 30 m/s and the "highest" color will be used when the particle speed exceeds 60 m/s. To show the default range at any time, click the **Compute** button and the **Min** and **Max** fields will be updated.

31.1.4.2.4. "Thinning" Pathlines

If your pathline plot is difficult to understand because there are too many paths displayed, you can "thin out" the pathlines by changing the **Path Skip** value in the *Pathlines Dialog Box* (p. 1975). By default, **Path Skip** is set to 0, indicating that a pathline will be drawn from each face on the selected surface (e.g., n pathlines). If you increase **Path Skip** to 1, every other pathline will be displayed, yielding $n/2$ pathlines. If you increase **Path Skip** to 2, every third pathline will be displayed, yielding $n/3$, and so on. The order of faces on the selected surface will determine which pathlines are skipped or drawn; thus adaption and reordering will change the appearance of the pathline display when a non-zero **Path Skip** value is used.

31.1.4.2.5. Coarsening Pathlines

To further simplify pathline plots, and reduce plotting time, a coarsening factor can be used to reduce the number of points that are plotted. Providing a coarsening factor of n, will result in each nth point being plotted for a given pathline in any cell. This coarsening factor is specified in the *Pathlines Dialog Box* (p. 1975), in the **Path Coarsen** field. For example, if the coarsening factor is set to 2, then ANSYS FLUENT will plot alternate points.

Important

Note that if any particle or pathline enters a new cell, this point will always be plotted.

31.1.4.2.6. Reversing the Pathlines

If you are interested in determining the source of a particle for which you know the final destination (e.g., a particle that leaves the domain through an exit boundary), you can reverse the pathlines and follow them from their destination back to their source. To do this, enable the **Reverse** option in the *Pathlines Dialog Box* (p. 1975). All other inputs for defining the pathlines will be exactly the same as for forward pathlines; the

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

only difference is that the surface(s) selected in the **Release From Surfaces** list will be the final destination of the particles instead of their source.

31.1.4.2.7. Plotting Oil-Flow Pathlines

If you want to display "oil-flow" pathlines (i.e., pathlines that are constrained to lie on a particular boundary), enable the **Oil Flow** option in the *Pathlines Dialog Box* (p. 1975). You will then need to select a single boundary zone in the **On Zone** list. The selected zone is the boundary on which the oil-flow pathlines will lie.

31.1.4.2.8. Controlling the Pulse Mode

If you are going to use the **Pulse** button in the *Pathlines Dialog Box* (p. 1975) to animate the pathlines, you can choose one of two pulse modes for the release of particles that follow the pathlines. To release a single wave of particles, select the **Single** option under **Pulse Mode**. To release particles continuously from the initial positions, select the **Continuous** option.

31.1.4.2.9. Controlling the Accuracy

If you are using large time step size for the calculation, there might be significant error introduced while calculating the pathlines. To control this error, select **Accuracy Control** and specify the value of **Tolerance**. The tolerance value will be taken in to consideration while calculating the pathlines for each time step.

31.1.4.2.10. Plotting Relative Pathlines

If you want to display the pathlines relative to the moving reference frame, enable the **Relative Pathlines** option in the *Pathlines Dialog Box* (p. 1975). You will then need to select the surfaces from the **Release from Surfaces** list.

31.1.4.2.11. Generating an XY Plot Along Pathline Trajectories

If you want to generate an XY plot along the trajectories of the pathlines you have defined, enable the **XY Plot** option in the *Pathlines Dialog Box* (p. 1975). The **Color by** drop-down list will be replaced by **Y Axis Function** and **X Axis Function** lists. Select the variable to be plotted on the y axis in the **Y Axis Function** list, and specify whether you want to plot this quantity as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory by selecting the appropriate item in the **X Axis Function** drop-down list. Specify the **Step Size**, number of **Steps**, and other parameters as usual for a standard pathline display. Then click **Plot** to display the XY plot.

Once you have generated an XY plot, you may want to save the plot data to a file. You can read this file into ANSYS FLUENT at a later time and plot it alone using the *File XY Plot Dialog Box* (p. 2016), as described in *XY Plots of File Data* (p. 1472), or add it to a plot of solution data, as described in *Including External Data in the Solution XY Plot* (p. 1472).

To save the plot data to a file, enable the **Write to File** option in the *Pathlines Dialog Box* (p. 1975). The **Plot** button will change to the **Write...** button. Clicking on the **Write...** button will open *The **Select File** Dialog Box* (p. 33), in which you can specify a name and save a file containing the plot data. The format of this file is described in *XY Plot File Format* (p. 1477).

31.1.4.2.12. Saving Pathline Data

To save pathline data to a file, perform the following steps:

1. Enable the **Write to File** option in the *Pathlines Dialog Box* (p. 1975) (*Figure 31.25* (p. 1402)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Type** drop-down list, select one of the following types of files:
   - **Standard** for **FIELDVIEW** (`.fvp`) format
   - **Geometry** for `.ibl` format (which can be read by **GAMBIT**)
   - **EnSight** format

   Important

   If you plan to write the pathline data in **EnSight** format, you should first verify that you have already written the files associated with the **EnSight Case Gold** file type by using the **File/Export...** menu option (see *EnSight Case Gold Files* (p. 91)).

   For further information about the files that are written for any of these types, please refer to the appropriate section following these steps.
3. Choose to color the pathlines by any of the scalar fields in the **Color by** drop-down lists.
4. Select the surface(s) from which to release the particles in the **Release From Surfaces** list.
5. If you selected **EnSight** under **Type**, you will need to specify the **EnSight Encas File Name**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** menu option. If you do not make a selection, then you will need to create an appropriate `.encas` file manually.

   You can also select the number of **Time Steps For EnSight Export**. This number directly determines how many time levels will be available for animation in EnSight.
6. Click on the **Write...** button to open *The **Select File** Dialog Box* (p. 33), in which you can specify a name and save a file containing the pathline data.

To initiate saving pathline data through the text command interface enter the following TUI command:

```
display/path-lines/write-to-files
```

In addition to pathline data, you can also export particle data in either **Standard**, **EnSight** or **Geometry** type. For information on exporting particle data in **FIELDVIEW** (standard), **EnSight** or `.ibl` (geometry) format, refer to *Exporting Steady-State Particle History Data* (p. 97).

31.1.4.2.12.1. Standard Type

If **Standard** is selected under **Type**, ANSYS FLUENT will write the file in **FIELDVIEW** format, which can be exported and read into **FIELDVIEW**. The **FIELDVIEW** ™ ASCII Particle Path Format is licensed from Intelligent Light, proprietor of an independent visualization software package (http://www.ilight.com). The file name that you use for saving the data must have a `.fvp` extension. You also have the ability to retrieve and display the particle and pathline trajectories from the file.

If the case is steady-state, the particle path information will be written in ASCII format. For transient or unsteady-state cases, the BINARY format must be used. The **FIELDVIEW** file contains a set of paths, where each path consists of a series of points. At every point the spatial location and selected variables are defined. A full description of the ASCII and BINARY formats can be found in Appendix K - Particle Path Formats of FIELDVIEW's Reference Manual [*1*] (p. 2209), available to licensed **FIELDVIEW** users.

The following is an example of the **FIELDVIEW** format for a steady-state case:

```
FVPARTICLES 2 1
Tag Names
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0
Variable Names
2
time
particle_id
3
0.2 0.8 1.3 0.2 0
0.3 0.9 1.3 0.4 0
0.5 1.1 1.3 0.6 0
```

The beginning of the file displays header information. `Tag Names` cannot be specified when the file is exported from ANSYS FLUENT, and hence will always be `0`. ANSYS FLUENT allows you to export two variables, which are listed under `Variable Names`: the first is determined by the scalar fields selected in the **Color by** drop-down lists (`time` in the example above); the second is always `particle_id`.

The rest of the file contains information about each path. A path section begins by listing the total number of points for the path. Then a line of data is presented for each point, with the X, Y, and Z locations listed in the first three columns and the variable information in the fourth and fifth columns. The example above presents a single pathline consisting of three points; the time ranges from `.2` to `.6`, and the ID of the particle is `0`.

31.1.4.2.12.2. Geometry Type

If **Geometry** is selected under **Type**, the file will be written in `.ibl` format. The resulting file contains particle paths in the form of a curve which can be read in GAMBIT. The following is an example of a **Geometry** file format that contains multiple curves:

```
Closed Index Arclength
Begin section ! 1
 Begin curve ! 1
 1                     185.61       0                    23.26
   2        88.90000000000001       0                   -89.67

   Begin curve ! 2
   1        88.89999999999569       0       -89.6699999999997
   2        76.90221619148909       0      -101.2290490001453
   3        62.92208239159677       0      -110.2907424975297
   4        47.47166726362848       0      -116.5231659809653
   5        31.11689338997181       0      -119.6980363161113
   6        14.45680848476821       0      -119.6990633707006
   7       -1.898356710978934       0      -116.5262095254603
   8       -17.34954014966171       0      -110.2956910520416
   9       -31.33079110697006       0      -101.2357213074894
   10        -43.330000000007       0      -89.67815166483965

   Begin curve ! 3
   1                    -43.33       0      -89.67815166485001
   2                   -175.56       0           64.69066040289
```

The above example demonstrates how multiple curves can be imported; single curves may also be imported. After importing this file into GAMBIT, the file is read by first looking for a `Begin curve` string and then looking for the X, Y, and Z coordinates under the `Begin curve` line.

31.1.4.2.12.3. EnSight Type

By selecting **EnSight** under **Type**, you can generate files with the following extensions:

- `.mpg`
- `.mscl`
- `.encas`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An `.mpg` file will be written for every time step specified in the **Time Steps For EnSight Export** field. A sequential number will be appended to the `.mpg` extension to indicate the time step. Each file contains a header which lists the time at which the data was exported, as well as three columns listing the X, Y and Z coordinates for every particle at that particular time step.

The following is an example of a file called `particle.mpg0003`, which contains data for nine particles at the third time step:

```
File is written from fluent in ensight measured particle format for
t = 2.42813e-04
particle coordinates
  9
  1-7.27734e-05 1.91710e-03 4.69093e-03
  2-1.75772e-04 1.97040e-03 3.92842e-03
  3-2.26051e-04 2.10134e-03 5.63228e-03
  4-1.16390e-04 2.32442e-03 5.23423e-03
  5-6.32735e-04 2.53326e-03 5.70791e-03
  6-9.69431e-04 2.37006e-03 5.27602e-03
  7-6.77868e-04 2.92054e-03 4.11570e-03
  8-9.78029e-04 2.75717e-03 4.13314e-03
  9-8.54859e-04 3.73727e-03 2.23796e-03
```

An `.mscl` file will be written for every time step specified in the **Time Steps For EnSight Export** field. A sequential number will be appended to the `.mscl`extension to indicate the time step. Each file contains the scalar information (specified under **Color By**) for every particle at a particular time step.

The following is an example of a file called `particle.mscl0006`, which captures **Particle ID** data for nine particles at the sixth time step:

```
particle id
 0.00000e+00 6.00000e+00 1.20000e+01 1.80000e+01 2.40000e+01 3.00000e+01
 3.60000e+01 4.20000e+01 4.80000e+01
```

A new `.encas` file will be written if a selection is made under **EnSight Encas File Name**. This new file is a modified version of the `.encas` file selected with the **Browse...** button, and contains information about all of the related files (including geometry, velocity, scalar and coordinate files). The name of the new file will be the root of the original file with `.new` appended to it (e.g. if `test.encas` is selected, a file named `test.new.encas` will be written). It is this new file that should be read into **EnSight**.

The following is an example of a file called `spray2-unsteady.new.encas`, that refers to the files generated when the data was originally exported as an **EnSight Case Gold** file type (`.geo`, `.vel`, `.sc11` and `.sc12`) and the files created during the pathline data export (`.mpg` and `.mscl`):

```
FORMAT
type: ensight gold
GEOMETRY
model: spray2-unsteady.geo
measured: 1 particle.mpg****
VARIABLE
scalar per measured node: 1 particle-id particle.mscl****
scalar per node: pressure                        spray2-unsteady.scl1
scalar per node: pressure-coefficient           spray2-unsteady.scl2
vector per node: velocity                        spray2-unsteady.vel
TIME
time set: 1 Model
number of steps: 10
filename start number:   1
filename increment:       1
time values: 0.00000e+00 1.21406e-04 2.42813e-04 3.64219e-04 4.85626e-04
6.07032e-04 7.28438e-04 8.49845e-04 9.71251e-04 1.09266e-03
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.4.2.13. Choosing Node or Cell Values

In ANSYS FLUENT you can determine the scalar field value at a particle location using the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are used. If you prefer to use the cell values, turn the **Node Values** option off. Note that for face-only functions like **Wall Shear Stress**, the cell value is the area-weighted average from the face values that define that cell as `c0`.

If you are plotting pathlines to show the effect of a porous medium or fan, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

31.1.5. Displaying Results on a Sweep Surface

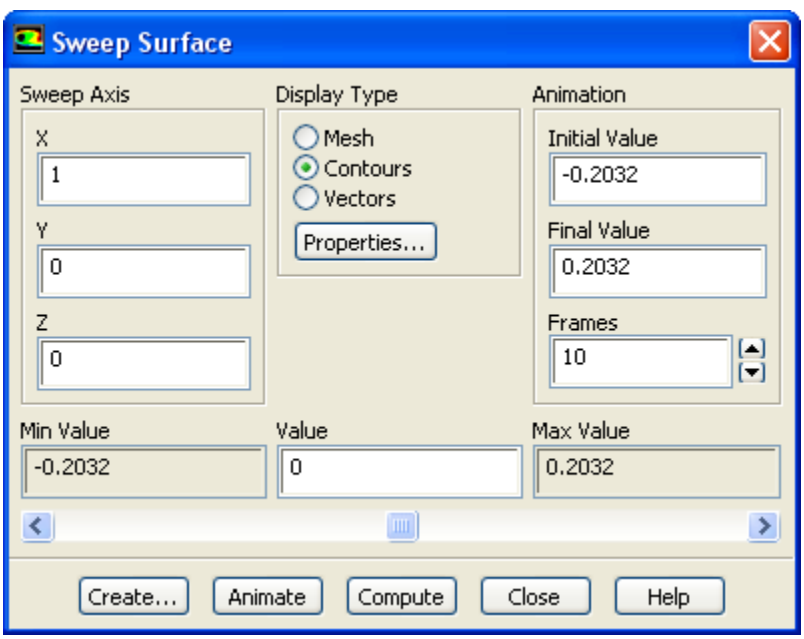
Sweep surfaces can be used when you want to examine the mesh, contours, or vectors on various sections of the domain without explicitly creating the corresponding surfaces. For example, if you want to display solution results for a 3D combustion chamber, instead of creating numerous surfaces at different cross-sections of the domain, you can use a sweep surface to view the variation of the flow and temperature throughout the chamber.

31.1.5.1. Steps for Generating a Plot Using a Sweep Surface

You can plot meshes, contours, or vectors on a sweep surface using the *Sweep Surface Dialog Box* (p. 1984) (*Figure 31.26* (p. 1409)).

Graphics and Animations → **Sweep Surface** → **Set Up...**

Figure 31.26 The Sweep Surface Dialog Box



The basic steps for generating a mesh, contour, or vector plot using a sweep surface are as follows:

1. Under **Sweep Axis**, specify the (**X**, **Y**, **Z**) vector representing the axis along which the surface should be swept.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Compute**to update the **Min Value** and **Max Value** to reflect the extents of the domain along the specified axis.
3. Under **Display Type**, specify the type of display you want to see: **Mesh**, **Contours**, or **Vectors**. The first time that you select **Contours** or **Vectors**, ANSYS FLUENT will open the *Contours Dialog Box* (p. 1967) or the *Vectors Dialog Box* (p. 1970) so you can modify the settings for the display. To make subsequent modifications to the display settings, click the **Properties...** button to open the *Contours Dialog Box* (p. 1967) or *Vectors Dialog Box* (p. 1970).
4. Move the slider under **Value** (which indicates the value of x, y, or z) to move the sweep surface through the domain along the specified **Sweep Axis**. ANSYS FLUENT will update the mesh, contour, or vector display when you release the slider. You can also enter a position in the **Value** field and press the `<Enter>` key to update the display.
5. If you want to save the currently displayed sweep surface so that you can use it for a different type of plot (e.g., a pathlines plot or an XY plot) or combine it with displays on other surfaces, click **Create...** to open the *Create Surface Dialog Box* (p. 1986) (*Figure 31.27* (p. 1410)). Enter the **Surface Name**and click **OK**.

   Figure 31.27 The Create Surface Dialog Box

   The surface that is created is an isosurface based on the mesh coordinates; the contour or vector settings are not stored in the surface.

You can also animate the sweep surface display, as described below, rather than moving the slide bar yourself.

31.1.5.2. Animating a Sweep Surface Display

The steps for animating a sweep surface display are as follows:

1. Specify the **Sweep Axis** and **Display Type** as described above.
2. Under **Animation**, enter the **Initial Value** and **Final Value** for the animation. These values correspond to the minimum and maximum values along the **Sweep Axis** for which you want to animate the display.
3. Specify the number of **Frames** you want to see in the animation.
4. Click **Animate**.

31.1.6. Hiding the Graphics Window Display

There may be situations where displaying graphics on a local machine is not practical, such as when running ANSYS FLUENT using the RSF (*Remote Simulation Facility (RSF)* (p. 13)). Therefore, you may decide to hide (or disable) the graphics display window.

To disable the graphics display window when starting ANSYS FLUENT from the command line, you can specify the driver as null:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
fluent -driver null
```

For an ANSYS FLUENT session that is already in progress, the graphics window display can be disabled using the following TUI command:

```
display → set → rendering-options → driver → null
```

Important

All graphics windows must be closed prior to invoking the above TUI command.

If the graphics window display is disabled, you can continue to save graphics using the **Save Picture** option, as described in *Saving Picture Files* (p. 117). The saved graphics files will be identical whether the graphics window display is enabled or disabled.

For an ANSYS FLUENT session that is already in progress, to re-enable a graphics window display that had been previously disabled, use the following TUI command:

```
display → set → rendering-options → driver → opengl
```

If any graphics windows are open (which are not visible to you), ANSYS FLUENT will prompt you to close all open windows. You can close them using the following Scheme command:

```
(close-all-open-windows)
```

and then retype the TUI command to enable the graphics windows.

Important

If you happen to be logged on to a machine remotely, then `opengl` may not work on your system. Use `x11` instead to enable your graphics windows.

31.2. Customizing the Graphics Display

There are a number of ways in which you can alter the graphical display once you have generated the basic elements in it (contours, meshes, etc.). For example, you can overlay multiple graphics, add descriptive text or lighting to the plot, and modify the captions or legend layout. These and other customizations are described in this section.

31.2.1. Overlay of Graphics
31.2.2. Opening Multiple Graphics Windows
31.2.3. Changing the Legend Display
31.2.4. Adding Text to the Graphics Window
31.2.5. Changing the Colormap
31.2.6. Adding Lights
31.2.7. Modifying the Rendering Options

31.2.1. Overlay of Graphics

Normally, you can see only one picture at a time in the graphics window; i.e., as one plot is generated, the previous plot is erased. Sometimes, however, you may want to see two plots overlaid. For example, you may want to plot vectors and pressure contours on the same plot (see *Figure 31.28* (p. 1412)). You can do this by turning on the **Overlays** option (and clicking **Apply**) in the *Scene Description Dialog Box* (p. 1994).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

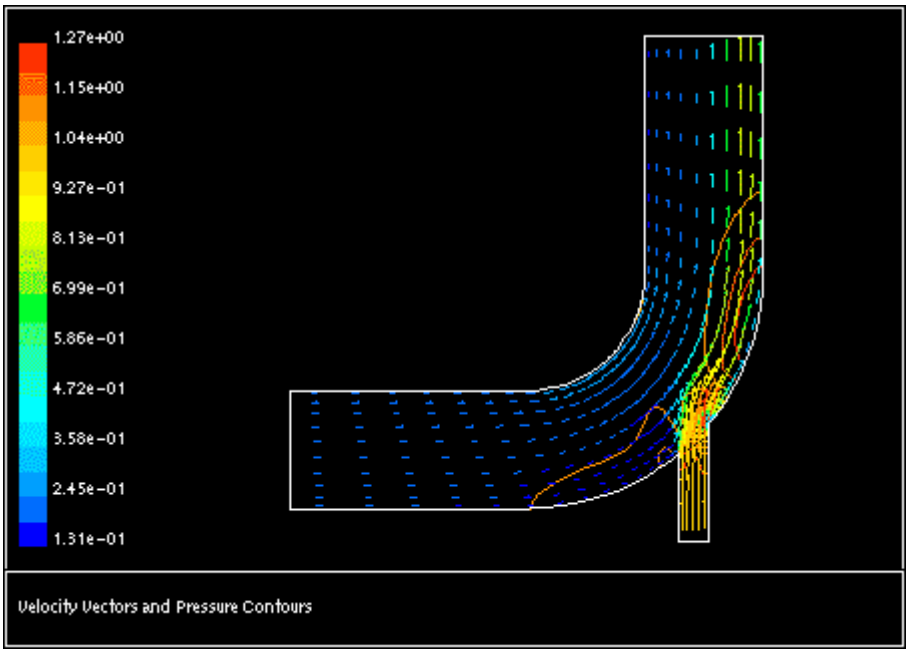
Graphics and Animations → **Scene...**

Once overlaying is enabled, subsequent graphics that you generate will be displayed on top of the existing display in the active graphics window. To generate a plot without overlays, you must turn off the **Overlays** option in the *Scene Description Dialog Box* (p. 1994) (and remember to click **Apply**).

When you are overlaying multiple graphics, the captions and color scale that will appear in the latest display are those that correspond to the most recently drawn graphic.

Note that when overlaying is enabled, it will apply to all graphics windows, including those that are not yet open. Turning overlays on and off does so for all graphics windows, not just for the active window. That is, if you enable overlays, open a new graphics window (as described in *Opening Multiple Graphics Windows* (p. 1412)), and then generate two or more graphics in that window, they will be overlaid.

Figure 31.28 Overlay of Velocity Vectors and Pressure Contours



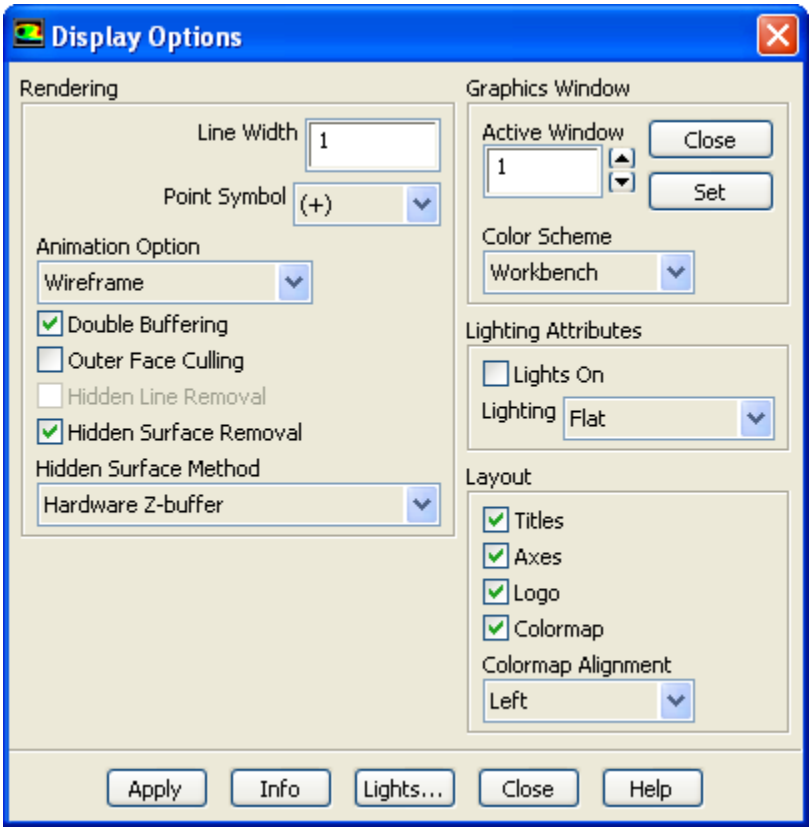
31.2.2. Opening Multiple Graphics Windows

During your ANSYS FLUENT session, you can open up to 20 graphics windows at one time and they may be viewed within the application window or in separate windows. The windows are numbered 1 through 20 and the ID number for each window will appear at the top of the frame that surrounds it. You can view a specific window by selecting it from the drop-down list next to the ID number. The first time you display graphics, window 1 will be displayed automatically. To open an additional window, you can use the *Display Options Dialog Box* (p. 1992) (*Figure 31.29* (p. 1413)).

Graphics and Animations → **Options...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.29 The Display Options Dialog Box



Use the up arrow to increment the window ID in the **Active Window** field under **Graphics Window** and then click the **Open** button. The **Close** button will change to the **Open** button if the **Active Window** is set to more than **1**.

To close an open window, increase or decrease the **Active Window** value to the ID of the window to be closed, and then click the **Close** button that appears next to the **Active Window** field. The **Open** button will change to the **Close** button if the **Active Window** is open.

To display a different color scheme for the graphics window, select **Classic** or **Workbench** from the drop-down list located below the **Active Window** field. All graphics windows will change.

31.2.2.1. Setting the Active Window

When you have more than one graphics window open, you must identify the active window so that ANSYS FLUENT will know which one to draw the plot in. There are two ways to set the active window: you can simply click any mouse button in the desired graphics window, or you can specify the ID for the desired graphics window in the **Active Window** field (in the *Display Options Dialog Box* (p. 1992)) and click the **Set** button. Regardless of the method used, this window will remain active until you set a new active window.

31.2.3. Changing the Legend Display

ANSYS FLUENT graphics include, by default, a caption or legend block that consists of fields of text describing the contents of the graphic, the ANSYS FLUENT product identification, an axis triad indicating the orientation of the displayed object, a color key defining the correspondence between each color and the magnitude of the plotted variable, and the ANSYS logo. You can turn off the display of the legend and color scale, and/or

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the axis triad. You can also hide the ANSYS logo. You can also display the colormap on any side of the display window as per convenience. In addition you can also edit the captions directly in the graphics window.

31.2.3.1. Enabling/Disabling the Legend, Logo, and Color Scale

You can disable the display of the **Titles**, **Axes**, **Logo**, and the **Colormap** scale by deselecting each of the options under **Layout** in the *Display Options Dialog Box* (p. 1992) (*Figure 31.29* (p. 1413)).

Graphics and Animations → **Options...**

Use the text interface to enable/disable the captions and color scale individually, and to change the size and position of the captions and color scale.

`display` → `set` → `windows` → `text` →

`display` → `set` → `windows` → `scale` →

See the separate Text Command List for details.

31.2.3.2. Editing the Legend

When captions are displayed in the graphics window, you may choose to modify, delete, or add to the text that appears in the caption box. To do so, click the left mouse button in the desired location. A cursor will appear, and you can then type new text or delete the text that was originally there (using the backspace or delete key). Note that changes to existing text in the caption block will be removed when you draw new graphics in the window (unless you are overlaying multiple graphics in the same window), but text that you add on a previously empty line in the caption block will not be removed until the default caption text makes use of that line.

31.2.3.3. Adding a Title to the Caption

You can define a title for your problem using the `title` text command:

`display` → `set` → `title`

The title you define will appear on the top line of the caption, at the far left, in all subsequent plots. It will also be saved in the case file.

Important

You will need to enclose your title in quotation marks (e.g., `"my title"`).

31.2.3.4. Enabling/Disabling the Axes

You can disable the display of the axis triad by turning off the **Axes** option under **Layout** in the *Display Options Dialog Box* (p. 1992) (*Figure 31.29* (p. 1413)).

31.2.3.5. Displaying/Hiding the Logo

You can prevent the ANSYS logo from being displayed in the graphics window by disabling the **Logo** option under **Layout** in the *Display Options Dialog Box* (p. 1992) (*Figure 31.29* (p. 1413)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

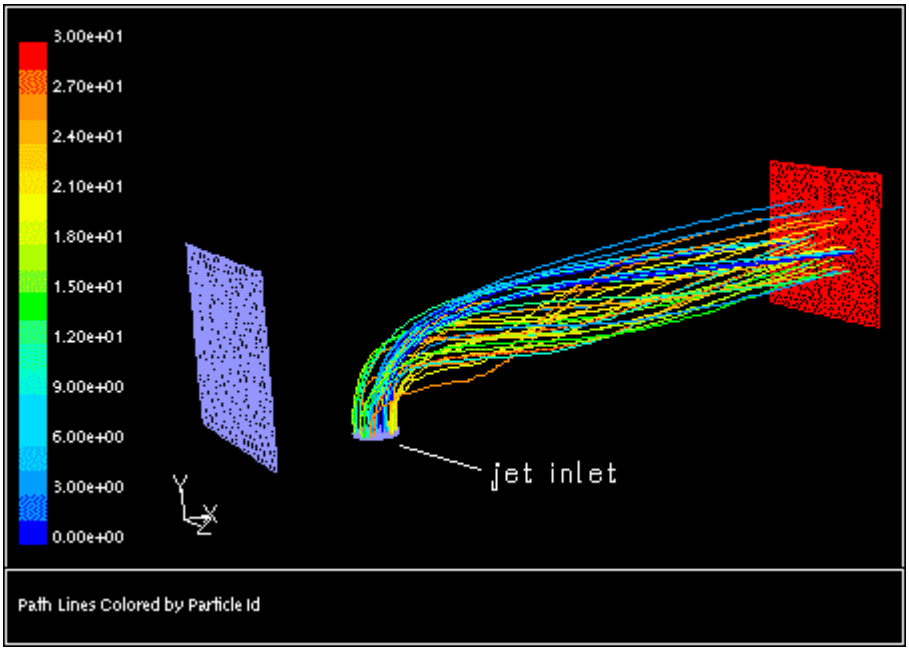
31.2.3.6. Colormap Alignment

You can set the position of the colormap on any side (left, top, bottom, or right) of the display window. Default alignment for the colormap is set to the **Left**. If you wish to change the alignment, select the required direction in the **Colormap Alignment** drop-down list.

31.2.4. Adding Text to the Graphics Window

There are two ways to add text annotations with optional attachment lines to the graphics windows. You can either use the **mouse-annotate** function for one of the mouse buttons, or use the *Annotate Dialog Box* (p. 2010). Both of these methods are described in this section. *Figure 31.30* (p. 1415) shows an example of a graphics display with annotated text in it.

Figure 31.30 Graphics Window with Text Annotation



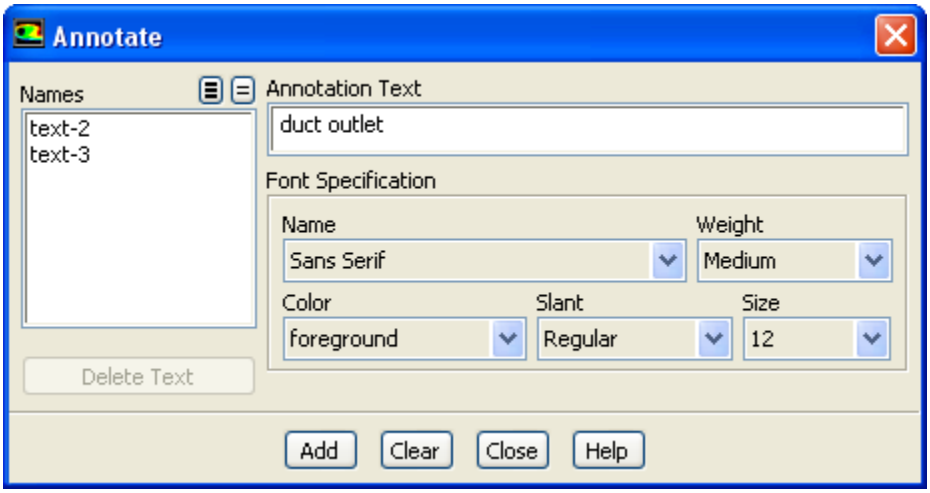
31.2.4.1. Adding Text Using the Annotate Dialog Box

Adding text to the graphics window using the *Annotate Dialog Box* (p. 2010) (*Figure 31.31* (p. 1416)) allows you to control the font and color of the text.

Graphics and Animations → Annotate...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.31 The Annotate Dialog Box



The steps for adding text are as follows:

1. Under **Font Specification**, select the font type in the **Name** drop-down list, the font weight (**Medium**or **Bold**) in the **Weight** drop-down list, the size (in points) in the **Size** drop-down list, the color in the **Color**drop-down list, and the slant (**Regular** or **Italic**) in the **Slant** drop-down list.
2. Enter the text to be added in the **Annotation Text** field.
3. Click **Add**. You will be asked to pick the location in the graphics window where you want to place the text, using the mouse-probe button. By default, the mouse-probe button is the right button, but you can change this using the *Mouse Buttons Dialog Box* (p. 2157), as described in *Controlling the Mouse Button Functions* (p. 1427).

   If you click the mouse button once in the desired location, the text will be placed at that point. Dragging the mouse with the mouse-probe button depressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The annotation text will be placed at the point where the mouse button was released.

31.2.4.2. Adding Text Using the Mouse-Annotate Function

To add text annotations to the graphics window using the **mouse-annotate** function, you must first set the function of one of the mouse buttons to be **mouse-annotate** in the *Mouse Buttons Dialog Box* (p. 2157). (See *Controlling the Mouse Button Functions* (p. 1427) for details about modifying the mouse button functions.) Then, click the **mouse-annotate** button in the desired location in the graphics window. A cursor will appear and you can type the text directly in the graphics window. Dragging the mouse with the **mouse-annotate** button depressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The cursor will then appear at the point where the mouse button was released.

Important

The mouse-annotate button will not function as described above when running ANSYS FLUENT on Linux/UNIX with the graphics window embedded. To work around this limitation, you can either annotate using the *Annotate Dialog Box* (p. 2010) (as described in *Adding Text Using the Annotate Dialog Box* (p. 1415)), or you can first detach your graphics window, so that it becomes a floating window, perform the annotation using the mouse-annotate button, then return to an embedded graphics window (as described in *Embedding the Graphics Windows* (p. 1432)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the *Annotate Dialog Box* (p. 2010) to edit or delete text added using the mouse, as described below.

31.2.4.3. Editing Existing Annotation Text

Once you have added text to the graphics display, using either the *Annotate Dialog Box* (p. 2010) or the **mouse-annotate** function, you may change the font characteristics of one or more text items, or delete individual text items.

To modify or delete existing text, follow these steps:

1. Select the appropriate item in the **Names** list in the *Annotate Dialog Box* (p. 2010) (*Figure 31.31* (p. 1416)). When you select a name, the associated text will be displayed in the **Annotation Text** field, and the **Add** button will become the **Edit** button.
2. Modify the **Font Specification** entries as desired, and click the **Edit** button to modify the text, or simply click the **Delete Text** button below the **Names** list to delete the selected text.

Note that if you want to make changes to all current annotation text, you can select all of the **Names** instead of just one in step 1.

You can move the text in the same way that you move other geometric objects in the display, using the *Scene Description Dialog Box* (p. 1994) and the *Transformations Dialog Box* (p. 1997). See *Transforming Geometric Objects in a Scene* (p. 1448) for details.

31.2.4.4. Clearing Annotation Text

Annotation text is associated with the active graphics window and is removed only when the annotations are explicitly cleared. To remove the annotations from the graphics window, you must click the **Clear** button in the *Annotate Dialog Box* (p. 2010) (even if you use the **mouse-annotate** function to add the text). If you draw new graphics in the window without clearing the annotations, they will remain visible in the new display.

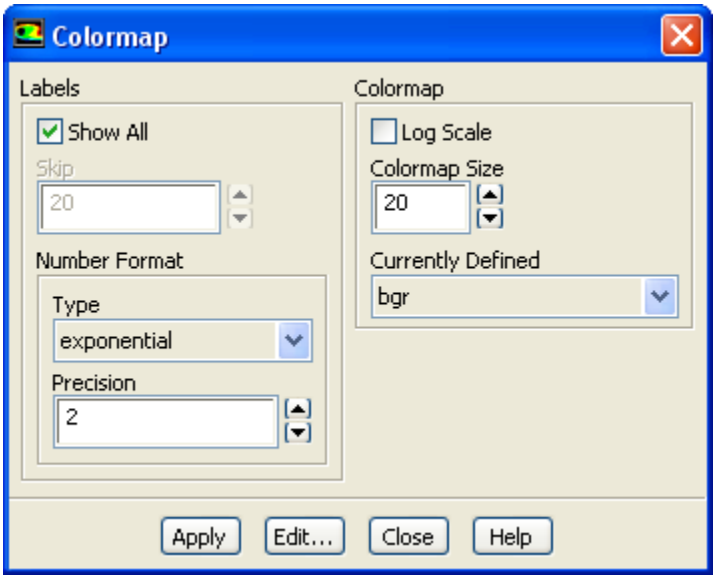
31.2.5. Changing the Colormap

The default colormap used by ANSYS FLUENT to display graphical data (e.g., vectors) ranges from blue (minimum value) to red (maximum value). Additional predefined colormaps are available, and you can also create custom colormaps. To make any changes to the colormap, you will use the *Colormap Dialog Box* (p. 2007) (*Figure 31.32* (p. 1418)).

Graphics and Animations → **Colormap**...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.32 The Colormap Dialog Box



When you plot contours, you can temporarily modify the number of colors in the colormap by changing the number of contour levels in the *Contours Dialog Box* (p. 1967); you will only need to use the *Colormap Dialog Box* (p. 2007) if you wish to change other characteristics of the colormap.

Important

Note that if you are using a gray-scale colormap and you wish to save a gray-scale picture, you should save a color picture. When you save a gray-scale picture, ANSYS FLUENT uses an internal gray scale, not the gray scale specified by the colormap. If you save a color picture, the colormap you selected (i.e., your gray scale) will be used.

31.2.5.1. Predefined Colormaps

The following colormaps are automatically available in ANSYS FLUENT:

bgr:
Blue represents the minimum value, green the middle, and red the maximum value. Colors in between are interpolated from blue to green, and from green to red. (This is the default colormap.)

bgrb:
Blue represents the minimum and maximum values, and green and red are values 1/3 and 2/3 of the maximum value, respectively. Colors in between are interpolated from blue to green, from green to red, and from red to blue.

blue:
The minimum value is represented by blue-black, and the maximum value by pure blue.

cyan-yellow:
Cyan represents the minimum value and yellow represents the maximum value.

fea:
Blue represents the minimum value and red represents the maximum value. The colors in between are those used in third-party finite element analysis packages.

gray:
Black is used for the minimum value and white for the maximum value.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

green:
The minimum value is represented by green-black, and the maximum value by pure green.

purple-magenta:
Purple represents the minimum value and magenta represents the maximum value.

red:
The minimum value is represented by red-black, and the maximum value by pure red.

rgb:
Red represents the minimum value, green the middle, and blue the maximum value. Colors in between are interpolated from red to green, and from green to blue.

The number of colors interpolated between the colors in the scale name (e.g., between purple and magenta) will depend on the size of the colormap.

31.2.5.2. Selecting a Colormap

The procedure for selecting a new colormap to be used in graphics displays is as follows:

1. In the *Colormap Dialog Box* (p. 2007) (*Figure 31.32* (p. 1418)), select the desired colormap in the **Currently Defined** drop-down list. This list will contain all of the colormaps predefined by ANSYS FLUENT as well as any custom colormaps that you have created as described in *Creating a Customized Colormap* (p. 1421).
2. Set the colormap size and scale as described in *Specifying the Colormap Size and Scale* (p. 1419).
3. Click **Apply** to update the current graphics display with the new colormap. All future displays will use the newly selected colormap and options.

31.2.5.2.1. Specifying the Colormap Size and Scale

Once you have selected the desired colormap from the **Currently Defined** list, you may modify the **Colormap Size**. This value is the number of distinct colors in the color scale.

You can also choose to use a logarithmic scale instead of a decimal scale by turning on the **Log Scale** option. With a log scale, the color used in the graphics display will represent the log of the value at that location in the domain. The values represented by the colors will, therefore, increase exponentially.

31.2.5.2.2. Changing the Number Format

You can change the format of the labels that define the color divisions at the left of the graphics window using the controls under the **Number Format** heading in the *Colormap Dialog Box* (p. 2007).

- To display the real value with an integral and fractional part (e.g., 1.0000), select **float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.
- To display the real value with a mantissa and exponent (e.g., 1.0e-02), select **exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.
- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **general** in the **Type** drop-down list.

31.2.5.3. Displaying Colormap Label

You can customize the number of values displayed on the colormap. The default number of labels that appear alongside the colormap depends on the font size and the colormap size (*Figure 31.33* (p. 1420)).

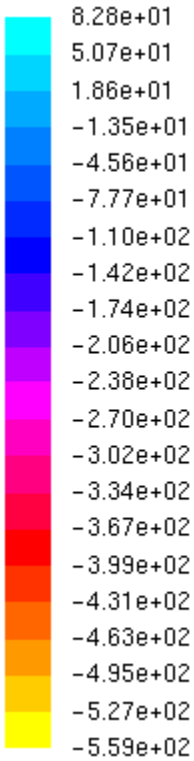
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To reduce the number of labels that appear alongside the colormap, increase the number of labels skipped. To do so, deselect **Show All** in the *Colormap Dialog Box* (p. 2007) and set the number of labels to be skipped.

  To demonstrate what effect this command has on the display, enter a value of 4 under **Skip** (note that the value entered must be an integer). This will result in three intermediate labels being skipped, with the first and the last colormap values always being displayed (*Figure 31.34* (p. 1421)).

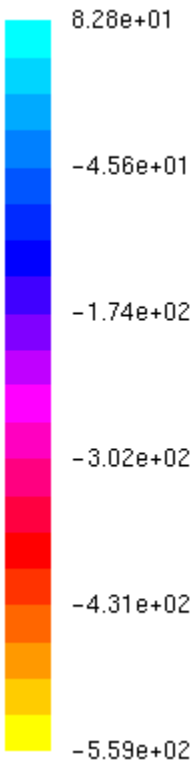
- To reset the original colormap display, simply select **Show All**.

Figure 31.33 The Default Colormap Label Display



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.34 The Colormap with Skipped Labels



31.2.5.4. Creating a Customized Colormap

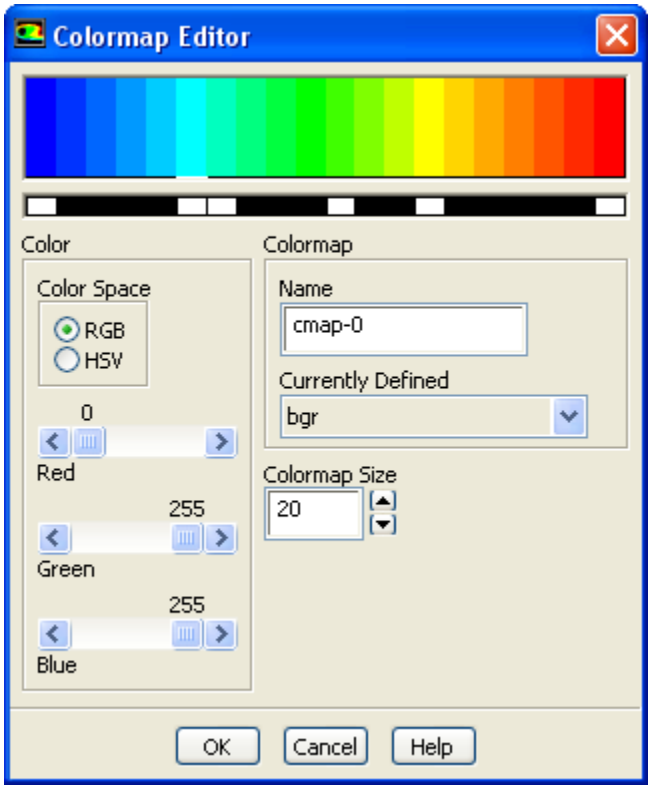
You can create your own colormap by manipulating the "anchor colors" and the colormap size. A color scale is created by linear interpolation between the anchor colors. The color, number, and position of the anchor colors will therefore control the description of the colormap. By increasing the colormap size, you can increase the total number of colors and obtain a color scale that changes more gradually.

The procedure you will follow is listed below:

1. In the *Colormap Dialog Box* (p. 2007), click the **Edit...** button to open the *Colormap Editor Dialog Box* (p. 2008) (*Figure 31.35* (p. 1422)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.35 The Colormap Editor Dialog Box



2. In the *Colormap Editor Dialog Box* (p. 2008), select a color scale in the **Currently Defined** list as your starting point. The colors in the scale will be displayed at the top of the dialog box. A white bar below a color is an "anchor point" indicating that this color is an "anchor color".

3. If you want to add more colors to the color scale, increase the **Colormap Size**; to use fewer colors, decrease this value. When you use the counter arrows (or type in a value and press <Enter>), the color scale display at the top of the dialog box will be updated immediately.

Important

The total number of colors must not be less than the number of anchor points.

4. To obtain the desired color scale interpolation, manipulate the anchor colors as needed:
   - To add an anchor point, click any mouse button on the black space directly below the desired anchor color (or click on the color itself). A white bar will appear below the color to identify it as an anchor color, and the color will automatically be selected for color-definition modification.
   - To remove an anchor point, click on the white bar below the anchor color. The white bar will disappear and the color scale will be updated to reflect the new interpolation.
   - To select a current anchor color in order to modify its color definition, click on the color itself at the top of the dialog box.
   - To modify the color of the selected anchor color, you can change either the red/green/blue components (choose **RGB**, the default) or the hue/saturation/value components (choose **HSV**). **HSV** is recommended if you plan to record the graphics display on video, as it allows you to create a more subtle gradation of color and reduce the tendency of bright colors to "bleed". Move the **Red**, **Green**,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **Blue** or **Hue**, **Saturation**, and **Value** sliders to obtain the desired color. The color scale at the top of the dialog box will be updated automatically to show the effect of your change.

Important

It is a good idea to note the original value of a color component before moving the slider so that you will be able to return to it if you change your mind. (See *Scales* (p. 31) for instructions on using a scale slider.)

If you make a mistake while modifying the color scale, you can start over by selecting the starting-point colormap in the **Currently Defined** list.

5. If you want to change the default name of the new colormap, enter the new name in the **Name** field. By default, custom colormaps are called `cmap-0`, `cmap-1`, etc.
6. Click **OK** to save the new colormap. The colormap name will now appear in the **Currently Defined** list in the *Colormap Dialog Box* (p. 2007) and can be selected for use in the graphics display.

Custom colormap definitions will be saved in the case file.

31.2.6. Adding Lights

In ANSYS FLUENT you can add lights with a specified color and direction to your display. These lights can enhance the appearance of the display when it contains 3D geometries. By default one light is defined. You can enable the effect of the existing light(s) using the *Display Options Dialog Box* (p. 1992) or the *Lights Dialog Box* (p. 2005), and you can add new lights using the *Lights Dialog Box* (p. 2005).

31.2.6.1. Turning on Lighting Effects with the Display Options Dialog Box

To enable the effect of lighting, you can use the *Display Options Dialog Box* (p. 1992).

Graphics and Animations → **Options...**

If you enable the **Lights On** option under **Lighting Attributes** and click the **Apply** button, you will see the lighting effects in the active graphics window. To turn off the lighting effects, simply turn off the **Lights On** option and click **Apply**.

You can also choose the method to be used in lighting interpolation; select **Flat**, **Gouraud**, or **Phong** in the **Lighting** drop-down list. Flat is the most basic method: there is no interpolation within the individual polygonal facets. Gouraud and Phong have smoother gradations of color because they interpolate on each facet.

31.2.6.2. Turning on Lighting Effects with the Lights Dialog Box

You can also enable lighting effects using the *Lights Dialog Box* (p. 2005) (*Figure 31.36* (p. 1424)).

Graphics and Animations → **Lights...**

For constant lighting effects in the direction of the view, enable the **Headlight On** option in the *Lights Dialog Box* (p. 2005). This option has the effect of a light source directly in front of the model, no matter what orientation the model is viewed in. To disable this feature, turn off the **Headlight On** option in the *Lights Dialog Box* (p. 2005).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

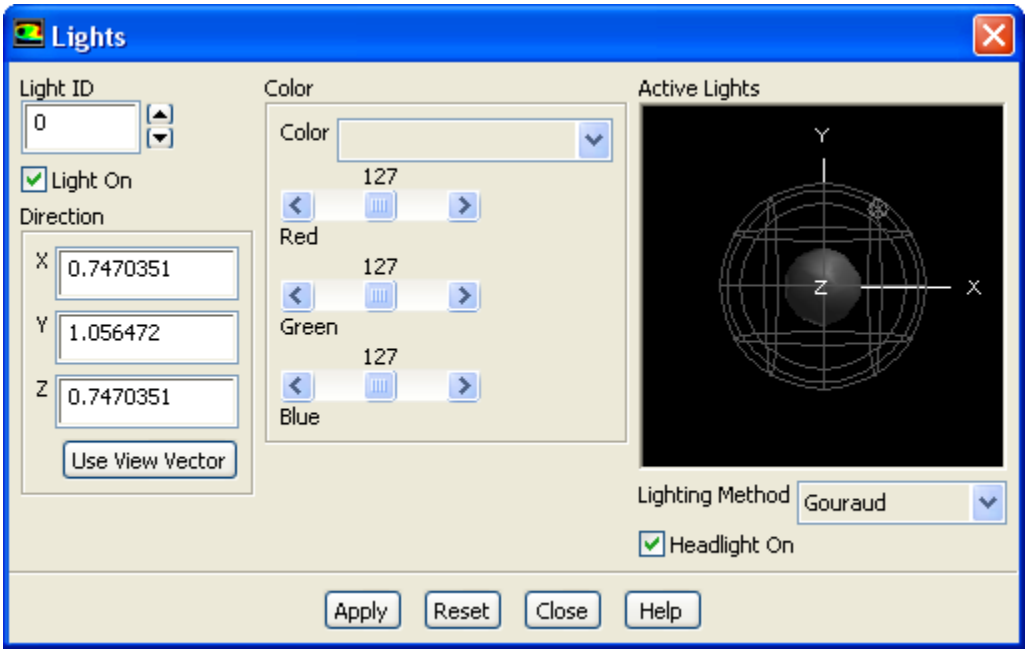
In the **Lighting Method** drop-down list, choose **Flat**, **Gouraud**, or **Phong** to enable the appropriate lighting method. These methods are described in the previous section. To disable lighting, select **Off** in the list. To see the lighting effects in the active graphics window, click the **Apply** button.

31.2.6.3. Defining Light Sources

You can control individual lights in the *Lights Dialog Box* (p. 2005) (*Figure 31.36* (p. 1424)). The *Lights Dialog Box* (p. 2005) allows you to create a light and then turn it off without deleting it. In this way, you can retain lights that you have defined previously but do not wish to use at present.

Graphics and Animations → **Lights...**

Figure 31.36 The Lights Dialog Box



(You can also open the *Lights Dialog Box* (p. 2005) by clicking on the **Lights...** button in the *Display Options Dialog Box* (p. 1992).)

By default, light 0 is defined to be dark gray with a direction of (1,1,1). A light source is a distant light, similar to the sun. The direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin. To create an additional light (e.g., light 1), follow the steps listed below.

1. Increase **Light ID** to a new value (e.g., `1`).
2. Enable the **Light On** check button.
3. Define the light color by entering a descriptive string (e.g., `lavender`) in the **Color** field, or by moving the **Red**, **Green**, and **Blue** sliders to obtain the desired color. The default color for all lights is dark gray.
4. Specify the light direction by doing one of the following:
   - Enter the (**X**, **Y**, **Z**) Cartesian components under **Direction**.
   - Click the middle mouse button in the desired location on the sphere under **Active Lights**. (You can also move the light along the circles on the surface of the sphere by dragging the mouse while

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

holding down the middle button.) You can rotate the sphere by pressing the left mouse button and moving the mouse (like a trackball).

- Use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. Then click the **Use View Vector** button to update the **X,Y,Z** fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

5. Repeat steps 1–4 to add more lights.
6. When you have defined all the lights you want, click **Apply** to save their definitions.

31.2.6.3.1. Removing a Light

To remove a light, enter the ID number of the light to be removed in the **Light ID** field and then turn off the **Light On** button. When a light is turned off, its definition is retained, so you can easily add it to the display again at a later time by turning on the **Light On** button. For example, you may want to define three different lights to be used in different scenes. You can define each of them, and then enable only one or two at a time, using the **Light ID** field and the **Light On** button. Once you have made all the desired modifications to the lights, remember to click the **Apply** button to save the changes.

31.2.6.3.2. Resetting the Light Definitions

If you have made changes to the light definitions, but you have not yet clicked on **Apply**, you can reset the lights by clicking on the **Reset** button. All lighting characteristics will revert to the last saved state (i.e., the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

31.2.7. Modifying the Rendering Options

Depending on the objects in your display window and what kind of graphics hardware and software you are using, you may want to modify some of the rendering parameters listed below. All are listed under the **Rendering** heading in the *Display Options Dialog Box* (p. 1992) (*Figure 31.29* (p. 1413)).

Graphics and Animations → Options...

After making a change to any of these rendering parameters, click the **Apply** button to re-render the scene in the active graphics window with the new attributes. To see the effect of the new attributes on another graphics window, you must redisplay it or make it the active window (see *Setting the Active Window* (p. 1413)) and click **Apply** again.

Line Width:
By default, all lines drawn in the display have a thickness of 1 pixel. If you want to increase the thickness of the lines, increase the value of **Line Width**.

Point Symbol:
By default, nodes displayed on surfaces and data points on line or rake surfaces are represented in the display by a + sign inside a circle. If you want to modify this representation (e.g., to make the nodes easier to see), you can select a different symbol in the **Point Symbol** drop-down list.

Animation Options:
There are two animation options which you can choose from. They are as follows:

All
uses a solid-tone shading representation of all geometry during mouse manipulation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wireframe
uses a wireframe representation of all geometry during mouse manipulation. If your computer has a graphics accelerator, you may not want to use this option; otherwise, the mouse manipulation may be very slow.

Double Buffering:
Enabling the **Double Buffering** option can dramatically reduce screen flicker during graphics updates. Note, however, that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Outer Face Culling:
This option allows you to turn off the display of outer faces in wall zones. **Outer Face Culling** is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you enable the **Outer Face Culling** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (i.e., walls with fluid or solid cells on both sides).

Hidden Line Removal:
If you do not use hidden line removal, ANSYS FLUENT will not try to determine which lines in the display are behind others; it will display all of them, and a cluttered display will result for most 3D mesh displays. For most 3D problems, therefore, you should enable the **Hidden Line Removal** option. You should turn this option off (for optimal performance) if you are working with a 2D problem or with geometries that do not overlap.

Hidden Surface Removal:
If you do not use hidden surface removal, ANSYS FLUENT will not try to determine which surfaces in the display are behind others; it will display all of them, and a cluttered display will result for most 3D mesh displays. For most 3D problems, therefore, you should enable the **Hidden Surface Removal** option. You should turn this option off (for optimal performance) if you are working with a 2D problem or with geometries that do not overlap.

You can choose one of the following methods for performing hidden surface removal in the **Hidden Surface Method** drop-down list. These options vary in speed and quality, depending on the device you are using.

Hardware Z-buffer
is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware-dependent. Note that if this method is not available on your computer, selecting it will cause the **Software Z-buffer** method to be used.

Painters
will show fewer edge-aliasing effects than **Hardware-Z-buffer**. This method is often used instead of **Software-Z-buffer** when memory is limited.

Software Z-buffer
is the fastest of the accurate software methods available (especially for complex scenes), but it is memory-intensive.

Z-sort only
is a fast software method, but it is not as accurate as **Software-Z-buffer**.

31.2.7.1. Graphics Device Information

If you need to know which graphics driver you are using and what graphics hardware it recognizes, you can click the **Info** button in the *Display Options Dialog Box* (p. 1992). The graphics device information will be printed in the text (console) window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.3. Controlling the Mouse Button Functions

A convenient feature of ANSYS FLUENT is that it allows you to assign a specific function to each of the mouse buttons. According to your specifications, clicking a mouse button in the graphics window will cause the appropriate action to be taken. These functions apply only to the graphics windows; they behave differently when an XY plot or histogram is displayed. For information about the use of mouse buttons in these plots, see *Plot Types* (p. 1466). Clicking any mouse button in a graphics window will make that window the active window.

Important

3DConnexion Space products (Ball, Mouse, Pilot, and Navigator) are not supported with ANSYS FLUENT.

For additional information, please see the following sections:

31.3.1. Button Functions
31.3.2. Modifying the Mouse Button Functions

31.3.1. Button Functions

The predefined button functions available are listed below:

mouse-rotate
allows you to rotate the view by dragging the mouse across the screen. Dragging horizontally rotates the object about the screen's y axis; vertical mouse movement rotates the object about the screen's x axis. The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-dolly
allows you to translate the view by dragging the mouse while holding down the button. The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-zoom
allows you to draw a zoom box, anchored at the point at which the button was pressed, by dragging the mouse with the button held down. When you release the button, if the dragging was from left to right, a magnified view of the area within the zoom box will fill the window. If the dragging was from right to left, the area of the window is shrunk to fit into the zoom box, resulting in a "zoomed out" view. If the mouse button is simply clicked (not dragged), the selected point becomes the center of the window.

mouse-roll-zoom
allows you to rotate or zoom the view, depending on the direction in which you drag the mouse. If you drag the mouse horizontally, the display will rotate about the axis normal to the screen. If you drag it vertically, the display will be magnified (if you drag it down) or shrunk (if you drag it up). The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-probe
allows you to select items from the graphics windows and request information about displayed scenes. If the probe function is turned off and you click the mouse-probe button in the graphics window, only the identity of the item on which you clicked will be printed out in the console window. If the probe function is turned on, more detailed information about a selected item will be printed out.

mouse-annotate
allows you to insert text into the graphics window. If the mouse button is dragged, an attachment line is drawn. When the button is released (after dragging or clicking), a cursor is displayed in the graphics window, and you can enter your text. When you are finished, press `<Enter>` or move the cursor out

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

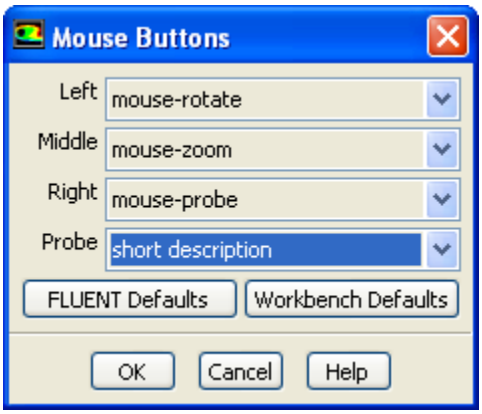
of the graphics window. To modify or remove annotated text and attachment lines, use the **Clear** button in the *Annotate Dialog Box* (p. 2010), as described in *Editing Existing Annotation Text* (p. 1417).

31.3.2. Modifying the Mouse Button Functions

Mouse button functions are specified in the *Mouse Buttons Dialog Box* (p. 2157) (*Figure 31.37* (p. 1428)).

Display → **Mouse Buttons...**

Figure 31.37 The Mouse Buttons Dialog Box



For each mouse button (**Left**, **Middle**, and **Right**), select the desired function in the drop-down list. The functions are listed above. If you assign the probe function to one of the buttons, select **on** or **off** as the **Probe** status.

The new button functions take effect as soon as you click **OK**. That is, you do not have to redraw the graphics window to use the new functions; the appropriate function will be executed when a mouse button is subsequently clicked in a graphics window.

The **ANSYS FLUENT Defaults** button functions are as follows:

Button	2D	3D
Left	**mouse-dolly**	**mouse-rotate**
Middle	**mouse-zoom**	**mouse-zoom**
Right	**mouse-probe**	**mouse-probe**

The **Workbench Defaults** button functions are as follows:

Button	2D	3D
Left	**mouse-dolly**	**mouse-dolly**
Middle	**mouse-rotate**	**mouse-probe**
Right	**mouse-zoom**	**mouse-zoom**

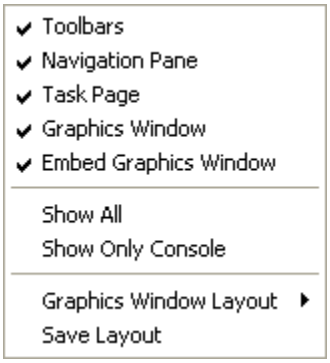
31.4. Viewing the Application Window

In ANSYS FLUENT, the application window will house the menus and console, as well as multiple graphics windows, task pages, and a navigation pane. By default, all components are displayed and one graphics window is visible. You can toggle the visibility of the toolbars, navigation pane, task page, and graphics

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

window. You can also detach or embed the graphics window. The menu bar and the console are never hidden. The graphics windows, when anchored within the application window, will be placed on the right side, immediately below the toolbar. You also have the option of viewing separate graphics windows as described in *Opening Multiple Graphics Windows* (p. 1412). A description of the **View** menu options which control the layout of the GUI is as follows.

Figure 31.38 The View Menu



Toolbars

allows you to customize the appearance of ANSYS FLUENT by displaying or hiding the ANSYS FLUENT toolbars. There are two toolbars. A general toolbar for read case, save and help commands. Another toolbar with commands that apply to the active graphics window. When the graphics window is hidden, the toolbars are visible. By default, all toolbars are visible.

Navigation Pane

allows you to customize the appearance of ANSYS FLUENT by displaying or hiding the **Navigation Pane** which is a small pane on the left side of the GUI. One row is always highlighted and signifies the task page that is displayed to the right of the **Navigation Page**. Depending on user input, sometimes one or more items are not displayed in the **Navigation Pane**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

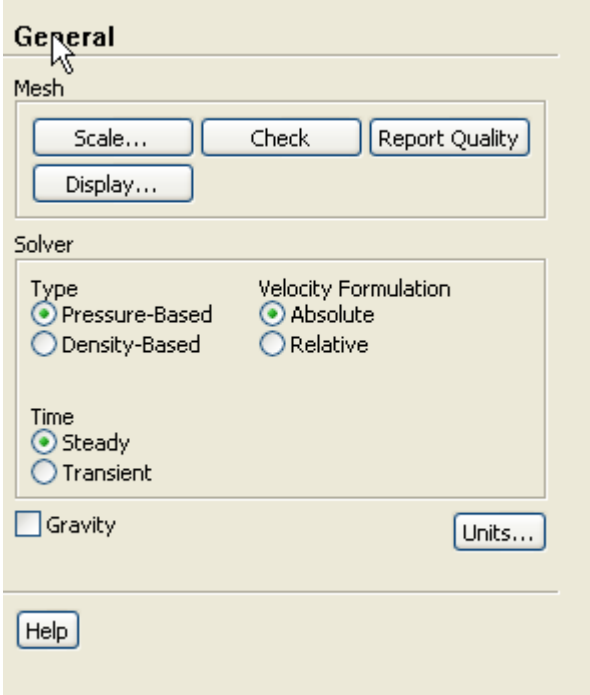
Figure 31.39 The Navigation Pane

Task Page

allows you to customize the appearance of ANSYS FLUENT by displaying or hiding the task page which is on the right side of the **Navigation Pane**. You can set the controls provided in the task page before running the calculation. The task page can also be opened using menu items. Each task page has a **Help** button which opens the help to the appropriate page in the reference guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.40 The General Task Page



Graphics Window

allows you to customize the appearance of ANSYS FLUENT by displaying or hiding the **Graphics Window** which is on the right side of the GUI, below the toolbar.

Embed Graphics Window

allows you to anchor the graphics window within ANSYS FLUENT, or detach the windows such that they are free-floating. For more information, refer to *Embedding the Graphics Windows* (p. 1432).

Show All

allows you to customize the appearance of ANSYS FLUENT by showing all of the GUI components with one command.

Show Only Console

allows you to customize the appearance of ANSYS FLUENT by showing only the console window along with the menu bar in the application window. The console window is anchored to the lower right corner of the application window. If **Show Only Console** is selected, the menu items of the hidden item (toolbars, navigation pane, task page, and graphics window) are unchecked in the **View** menu. You can resize the console window as you wish with the minimum height being 2 lines. The console window will display messages to the user and provide access to the TUI.

Important

Whenever a GUI component is hidden and you issue a command that requires a hidden GUI component, the view is changed automatically to complete the request.

Graphics Window Layout

allows you to customize the appearance of ANSYS FLUENT by displaying multiple views. This menu has a submenu with several options. By default, one graphics window is displayed and the window numbering will start at 1. As you can see in *Figure 31.42* (p. 1432), you may display up to 4 windows. You can select any existing window to view through the drop-down menu above the window. You may click on any window to change it from inactive to active.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.41 Graphics Window Layout Submenu

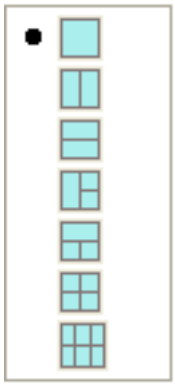
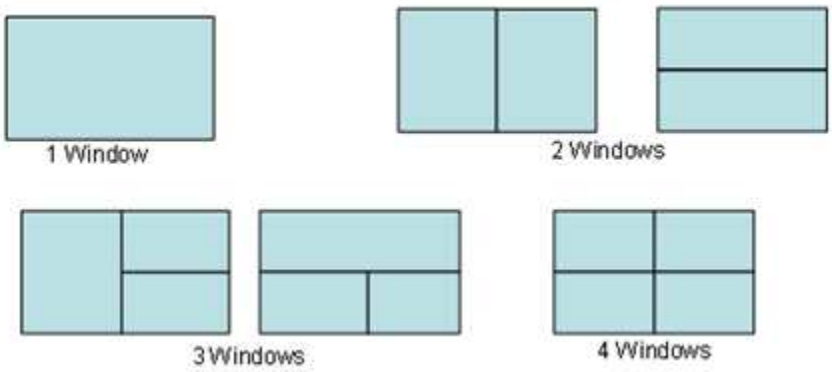


Figure 31.42 Graphics Window Layout Options



Save Layout

allows you to save the current layout of the GUI, including the visibility of the current GUI components and the configuration of the dialog boxes and graphics window. This saved layout is applied when you start ANSYS FLUENT, utilizing a `.cxlayout` file that is written in your home directory. The default position will be used for any dialog box that is not repositioned when you save the layout. The `.cxlayout` file in your home directory applies to all Cortex applications (i.e., ANSYS FLUENT, MixSim, and TGrid).

31.4.1. Embedding the Graphics Windows

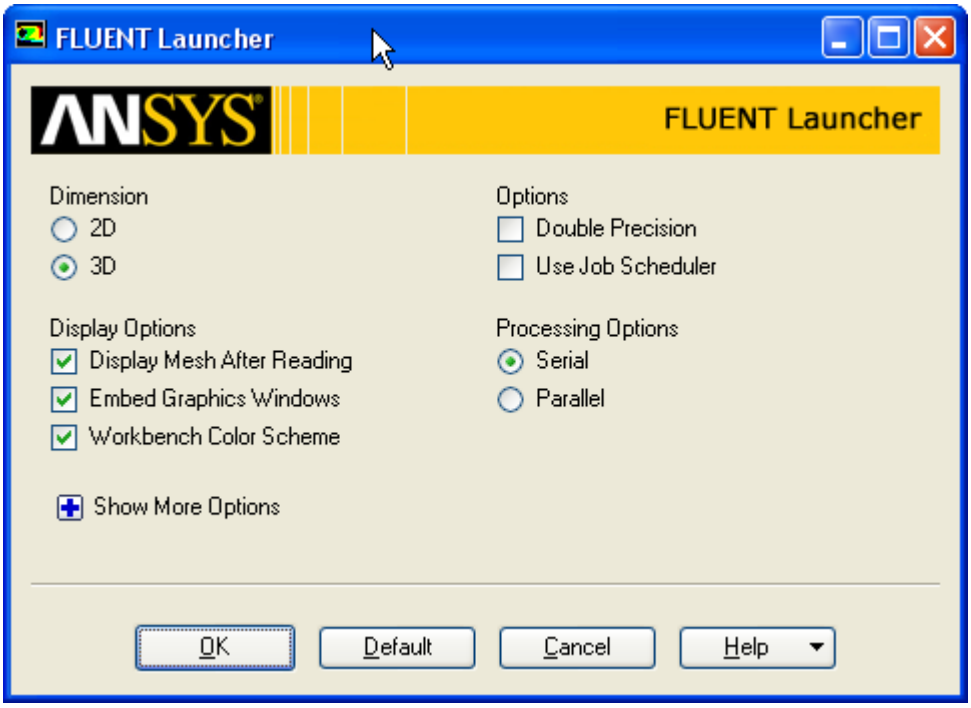
When starting ANSYS FLUENT, FLUENT Launcher has an option which allows you to embed the graphics window in the main application or show them as separate windows. The **Embed Graphics Window** option under **Display Options** is enabled by default.

Important

Note that FLUENT Launcher saves your most recent settings. Therefore, if this option was previously disabled, then it will remain disabled the next time you use FLUENT Launcher.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.43 The FLUENT Launcher Dialog Box



If you started ANSYS FLUENT with the **Embed Graphics Window** option disabled, and you do not want to have separate, floating graphics windows, you can always embed the graphics windows again. To do so, use the **View/Embed Graphics Window** menu item.

31.5. Modifying the View

ANSYS FLUENT allows you to select and control the view of the scene that is displayed in the graphics window. You can modify the view by scaling, centering, rotating, translating, or zooming the display. You can also save a view that you have created, or restore or delete a view that you saved earlier. These operations are performed in the *Views Dialog Box* (p. 2000) (*Figure 31.45* (p. 1435)) or with the mouse, or simply using the shortcut in the graphics toolbar, as described in *Selecting a View* (p. 1434).

Important

You can revert to the previous view by pressing the `<Ctrl>` and `L` keys while the graphics window has the main focus. You can also use the text command `view last` from the top level of the text command tree. You can use the command to revert to any of the past 20 views.

For additional information, please see the following sections:

31.5.1. Selecting a View
31.5.2. Manipulating the Display
31.5.3. Controlling Perspective and Camera Parameters
31.5.4. Saving and Restoring Views
31.5.5. Mirroring and Periodic Repeats

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.1. Selecting a View

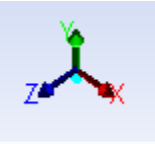
You can use the *Views Dialog Box* (p. 2000) (*Figure 31.45* (p. 1435)) to select the orientation of your display or use the drop-down available in the graphics toolbar for 3D simulations. Furthermore, you can simply click on the interactive triad (*Figure 31.44* (p. 1434)), displayed in the graphics window. The *Table 31.1: Standard Views* (p. 1434) relates the **Views** listed in the *Views Dialog Box* (p. 2000) to the views available in the graphics toolbar (see *The Graphics Toolbar* (p. 23)).

Graphics and Animations → Views...

Table 31.1 Standard Views

View Listed in Dialog Box	View Shortcut in Graphics Toolbar
back	
bottom	
front	
isometric	
left	
right	
top	

Figure 31.44 Using the Triad to Change the Orientation of the Object



To change the orientation of the object in the graphics window using the triad (*Figure 31.44* (p. 1434)), you can

- Left-click on an axis to point it in the positive direction.
- Right-click on an axis to point it in the negative direction.
- Left-click on the iso-ball to set the isometric view.

Note that left-clicking on X-axis will result in the +X-axis pointing towards you.

31.5.2. Manipulating the Display

Most of the manipulating activities (like scaling and centering the display) are accomplished using the *Views Dialog Box* (p. 2000) (*Figure 31.45* (p. 1435)).

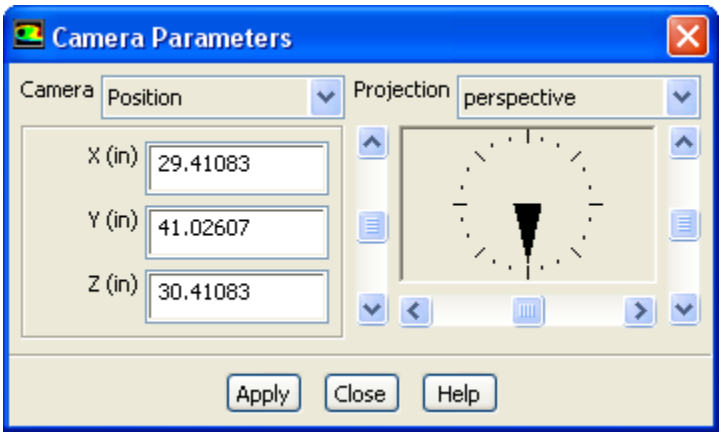
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graphics and Animations → Views...

Figure 31.45 The Views Dialog Box

You can rotate, translate, and zoom the graphics display using either the mouse or the *Camera Parameters Dialog Box* (p. 2005) (*Figure 31.46* (p. 1435)), which is opened from the *Views Dialog Box* (p. 2000).

Figure 31.46 The Camera Parameters Dialog Box



31.5.2.1. Scaling and Centering

You can scale and center the current display without changing its orientation by clicking on the **Auto Scale** button in the *Views Dialog Box* (p. 2000).

31.5.2.2. Rotating the Display

Use either the mouse or the *Camera Parameters Dialog Box* (p. 2005) to:

- Rotate the display in any direction (in 3D)
- Rotate the display about the axis normal to the screen (in 2D)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To rotate a display with the mouse, use the button with the **mouse-rotate** function (the left button, by default). For information about changing the mouse functions, see *Controlling the Mouse Button Functions* (p. 1427).

- Click and drag the left mouse button in the graphics window to rotate the geometry in the display.
- You can also click and drag the left mouse button on the (x, y, z) graphics triad in the lower left corner to rotate the display.
- If you press the `<Shift>` key when you first click the mouse button to begin the rotation, the rotation will be constrained to a single direction (e.g., you can rotate about the screen's horizontal axis without changing the position relative to the vertical axis).
- If you want to constrain the rotation of a display to be about the axis normal to the screen, you can also use the **mouse-roll-zoom** function.
- Click the appropriate mouse button and drag the mouse to the left for clockwise rotation, or to the right for counter-clockwise rotation.

To rotate a 3D display using the *Camera Parameters Dialog Box* (p. 2005) (*Figure 31.46* (p. 1435)) use the dial and the slider of the scales.

- To rotate about the horizontal axis at the center of the screen, move the slider on the scale to the left of the dial up or down (see *Scales* (p. 31) or instructions on using the scale).
- To rotate about the vertical axis at the center of the screen, move the slider on the scale below the dial to the left or right.
- To rotate about the axis at the center of and perpendicular to the screen, click the left mouse button on the indicator in the dial and drag it around the dial.

Important

The position of the slider or the dial indicator does not reflect the cumulative rotation about the axis as the slider/indicator will return to its original position when you release the mouse button.

31.5.2.2.1. Spinning the Display with the Mouse

When you use the mouse for rotation, you have the option to "push" the display into a continuous spin. This feature can be used in conjunction with video recording, or simply for interactive viewing of the domain from different angles. To activate this option, use the `auto-spin?` text command:

`display` → `set` → `rendering-options` → `auto-spin?`

Then display the graphics (or, if the graphics are already displayed, you can click the **Apply** button in the *Display Options Dialog Box* (p. 1992)). The **mouse-rotate** button will then have two uses:

- To perform the standard rotation, stop dragging the mouse before you release the **mouse-rotate** button.
- To start the continuous spin, release the **mouse-rotate** button while you are still dragging the mouse. The display will continue to rotate on its own until you click any mouse button in the graphics window again. The speed of the rotation will depend on how fast you are dragging the mouse when you release the button.

For smoother rotation, enable the **Double Buffering** option in the *Display Options Dialog Box* (p. 1992) (see *Modifying the Rendering Options* (p. 1425)). This will reduce screen flicker during graphics updates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.2.3. Translating the Display

By default the left mouse button is set to **mouse-dolly** in 2D. For information about changing the mouse functions, see *Controlling the Mouse Button Functions* (p. 1427). Click and drag the left mouse button in the graphics window to translate the geometry in the display.

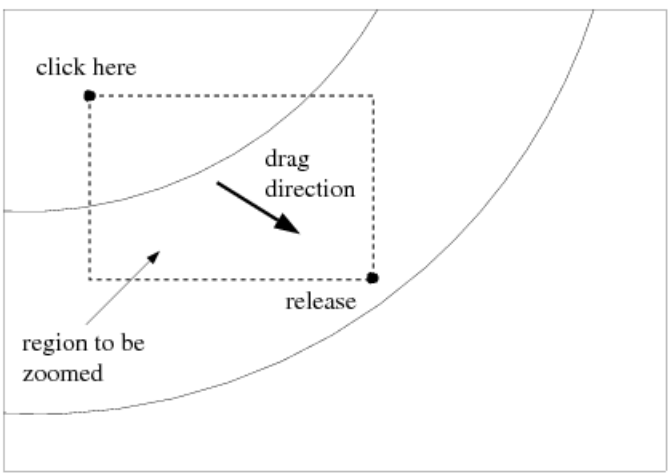
In 3D, you can either change one of the button functions to **mouse-dolly** and follow the instructions above for 2D, or use the **mouse-zoom** button (the middle button by default). Click the middle button once on the point in the display that you want to move to the center of the screen. ANSYS FLUENT will redisplay the graphic with that point in the center of the window. This method can also be used in 2D.

31.5.2.4. Zooming the Display

In both 2D and 3D you will use the mouse button with the **mouse-zoom** function (the middle button by default) or the **mouse-roll-zoom** function (see *Controlling the Mouse Button Functions* (p. 1427) for information about enabling this optional function), or the *Camera Parameters Dialog Box* (p. 2005) to magnify and shrink the display.

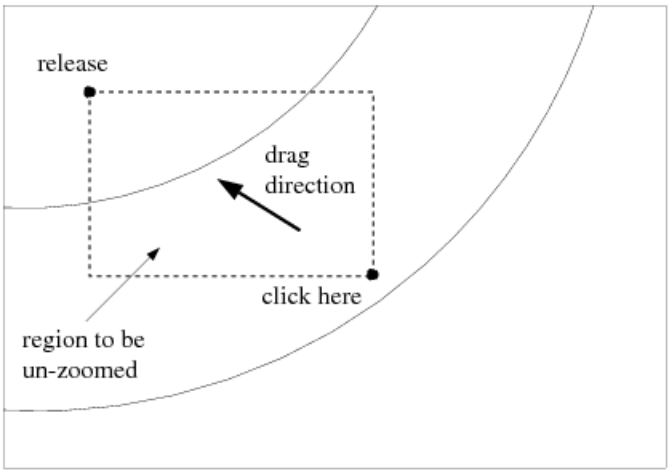
With the **mouse-zoom** function, click the middle mouse button and drag it from left to right (creating a "zoom box") to magnify the display. *Figure 31.47* (p. 1437) displays the correct dragging of the mouse, from upper left to lower right on the display, in order to zoom. You can also drag from lower left to upper right. After you release the mouse button, ANSYS FLUENT will redisplay the graphic, filling the graphics window with the portion of the display that previously occupied the zoom box.

Figure 31.47 Zooming In (Magnifying the Display)



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.48 Zooming Out (Shrinking the Display)



Click the middle mouse button and drag it from right to left to shrink the display. *Figure 31.48* (p. 1438) displays the correct dragging of the mouse, from lower right to upper left on the display, in order to "zoom out". You can also drag from upper right to lower left. After you release the mouse button, ANSYS FLUENT will redisplay the graphic, shrinking the graphical display by the ratio of sizes of the zoom box you created and the previous display.

With the **mouse-roll-zoom** function, click the appropriate mouse button and drag the mouse down to zoom in continuously, or up to zoom out. In the *Camera Parameters Dialog Box* (p. 2005) (*Figure 31.46* (p. 1435)), use the scale to the right of the dial to zoom the display. Move the slider bar up to zoom in and down to zoom out.

31.5.3. Controlling Perspective and Camera Parameters

Perspective and other camera parameters are defined in the *Views Dialog Box* (p. 2000), which you can open by clicking on the **Camera...** button in the *Camera Parameters Dialog Box* (p. 2005) (*Figure 31.45* (p. 1435)).

Graphics and Animations → **Views...**

31.5.3.1. Perspective and Orthographic Views

You may choose to display either an orthographic view or a perspective view of your graphics. To show a perspective view (the default), select **Perspective** in the **Projection** drop-down list in the *Views Dialog Box* (p. 2000) (*Figure 31.46* (p. 1435)). To turn off perspective, select **Orthographic** in the **Projection** drop-down list.

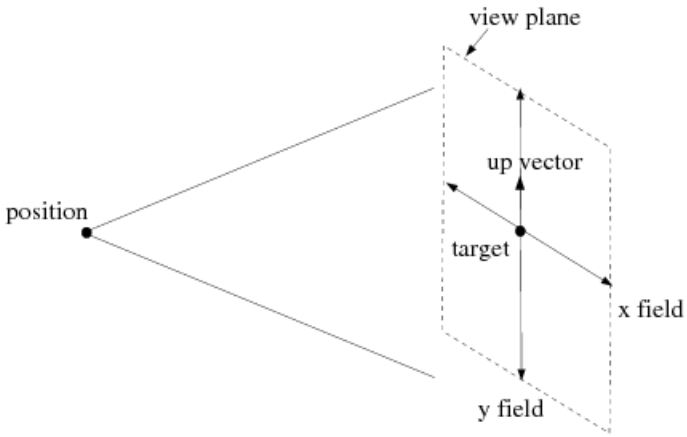
31.5.3.2. Modifying Camera Parameters

Instead of translating, rotating, and zooming the display as described in *Manipulating the Display* (p. 1434), you may sometimes want to modify the "camera" through which you are viewing the graphics display.

The camera is defined by four parameters: position, target, up vector, and field, as illustrated in *Figure 31.49* (p. 1439). "Position" is the camera's location. "Target" is the location of the point the camera is looking at, and "up vector" indicates to the camera which way is up. "Field" indicates the field of view (width and height) of the display.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.49 Camera Definition



To modify the camera's position, select **Position** in the **Camera** drop-down list and specify the **X**, **Y**, and **Z** coordinates of the desired point. To modify the target location, select **Target** in the **Camera** drop-down list and specify the coordinates of the desired point. Select **Up Vector** to change the up direction. The up vector is the vector from (0,0,0) to the specified (**X,Y,Z**) point. Finally, to change the field of view, select **Field** in the **Camera** list and enter the **X** (horizontal) and **Y** (vertical) field distances.

Important

Click **Apply** after you change each camera parameter (**Position**, **Target**, **Up Vector**, and **Field**).

31.5.4. Saving and Restoring Views

After you make changes to the view shown in your graphics window, you may want to save the view so that you can return to it later. Several default views are predefined for you, and can be easily restored. All saving and restoring functions are performed with the *Views Dialog Box* (p. 2000) (*Figure 31.45* (p. 1435)).

Graphics and Animations → Views...

Important

Note that settings for mirroring and periodic repeats are not saved in a view.

31.5.4.1. Restoring the Default View

When experimenting with different view manipulation techniques, you may accidentally "lose" your geometry in the display. You can easily return to the default (front) view by clicking the **Default** button in the *Views Dialog Box* (p. 2000).

31.5.4.2. Returning to Previous Views

After manipulating the display and viewing it from different angles, you can return to previous displays by clicking the **Previous** button in the *Views Dialog Box* (p. 2000).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.4.3. Saving Views

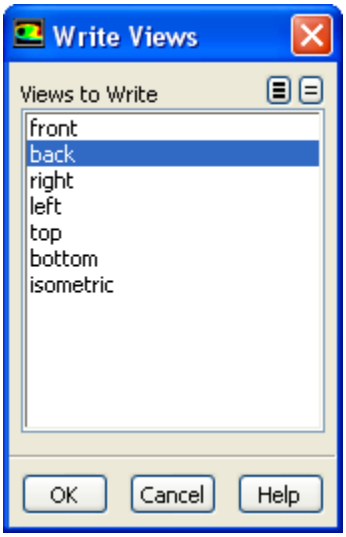
Once you have created a new view that you want to save for future use, enter a name for it in the **Save Name** field in the *Views Dialog Box* (p. 2000) and click the **Save** button. Your new view will be added to the list of **Views**, and you can restore it later as described below.

If a view with the same name already exists, you will be asked in a *Question Dialog Box* (p. 32) if it is OK to overwrite the existing view. If you overwrite one of the default views (top, left, right, front, etc.), be sure to save it in a view file if you wish to use it in a later session. Although all views are saved to the case file, the default views are recomputed automatically when a case file is read in. Any custom view with the same name as a default view will be overwritten at that time.

As mentioned previously, all defined views will be saved in the case file when you write one. If you plan to use your views with another case file, you can write a "view file" containing just the views. You can read this view file into another solver session involving a different case file and restore any of the defined views, as described below.

To save a view file, click the **Write...** button in the *Views Dialog Box* (p. 2000). In the resulting *Write Views Dialog Box* (p. 2002) (*Figure 31.50* (p. 1440)), select the views you want to save in the **Views to Write** list and click **OK**. You will then use the *The **Select File** Dialog Box* (p. 33) to specify the file name and save the view file.

Figure 31.50 The Write Views Dialog Box



31.5.4.4. Reading View Files

If you have saved views to a view file (as described above), you can read them into your current solver session by clicking on the **Read...** button in the *Views Dialog Box* (p. 2000), and indicating the name of the view file in *The **Select File** Dialog Box* (p. 33). If a view that you read has the same name as a view that already exists, you will be asked in a *Question Dialog Box* (p. 32) if it is OK to overwrite (i.e., replace) the existing view.

31.5.4.5. Deleting Views

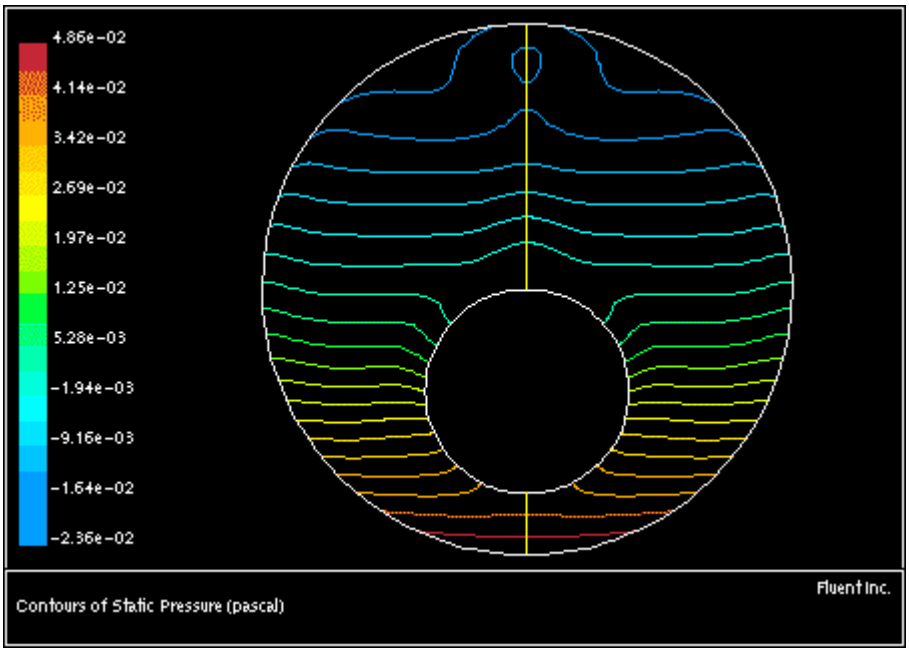
If you decide that you no longer want to keep a particular view, you can delete it by selecting it in the **Views** list and clicking on the **Delete** button. Use this option carefully, so that you do not accidentally delete one of the predefined views.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.5. Mirroring and Periodic Repeats

If you model the problem domain as a subset of the complete geometry using symmetry or periodic boundaries, you can display results on the complete geometry by mirroring or repeating the domain. For example, only one half of the annulus shown in *Figure 31.51* (p. 1441) was modeled, but the graphics are displayed on both halves. You can also define mirror planes or periodic repeats just for graphical display, even if you did not model your problem using symmetry or periodic boundaries.

Figure 31.51 Mirroring Across a Symmetry Boundary



Display of symmetry and periodic repeats is controlled in the *Views Dialog Box* (p. 2000) (*Figure 31.52* (p. 1441)).

Graphics and Animations → Views...

Figure 31.52 The Views Dialog Box

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a symmetric domain, all symmetry boundaries are listed in the **Mirror Planes** list. Select one or more of these boundaries as the plane(s) about which to mirror the display.

For a periodic domain, click the **Define...** button to open the *Graphics Periodicity Dialog Box* (p. 2003), to access the periodicity parameters. Specify the number of times to repeat the modeled portion by increasing the value of **Number of Repeats**. If, for example, you modeled a 90° sector of a duct and you wanted to display results on the entire duct, you would set **Number of Repeats** to 4.

In some cases, there may be multiple zones with different periodicity in the domain. For example, in turbomachinery problems with multiple blade rows using the mixing plane model, the periodic angles are different for each blade row. One blade may contain 20 blades (18° periodic angle) and other may contain 15 blades (24° periodic angle). In such cases select the required cell zone and specify the number of repeats for that particular cell zone.

When you click **Set** in the *Graphics Periodicity Dialog Box* (p. 2003) the graphics display will be immediately updated to show the requested periodic repeats.

Figure 31.53 (p. 1442) and *Figure 31.54* (p. 1442) shows the display for the sample geometry before and after applying the periodic repeats respectively. In this case the value of **Number of Repeats** is set to 6 for the 60° sector (outer part) and to a value of 4 is set for the 90° sector (inner part) of the geometry.

Figure 31.53 Before Applying Periodicity

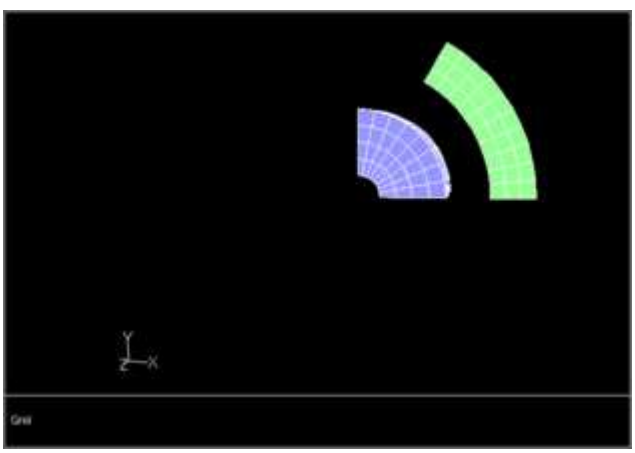
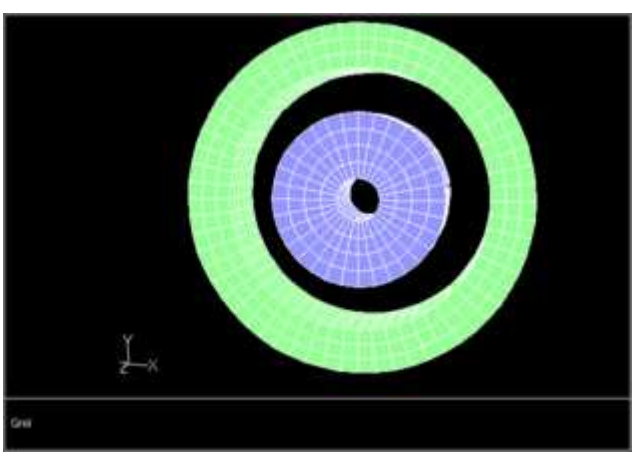


Figure 31.54 After Applying Periodicity



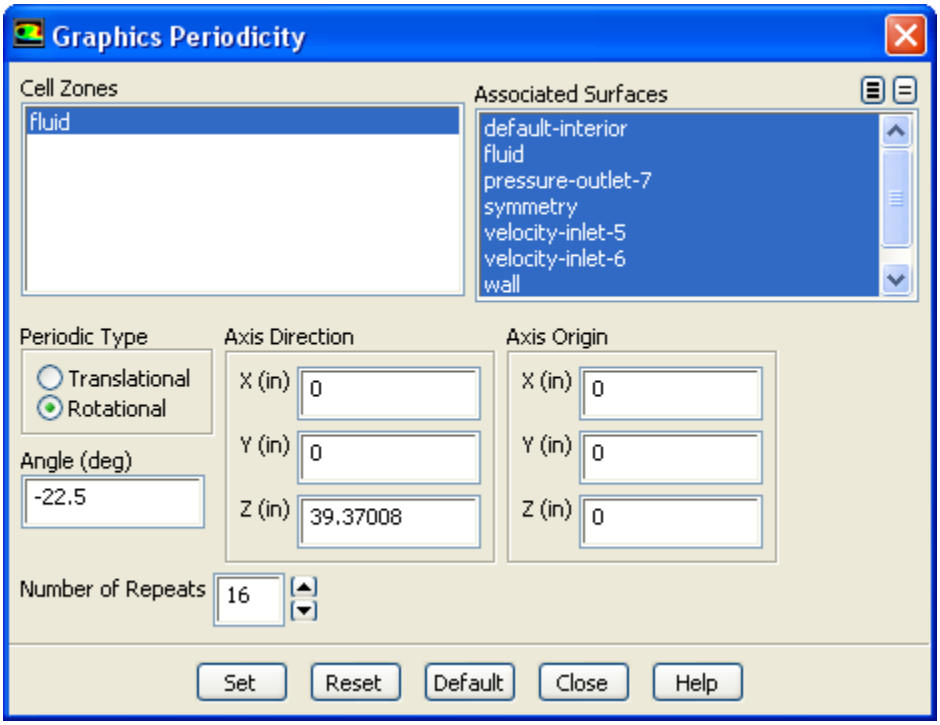
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.5.1. Periodic Repeats for Graphics

To define graphical periodicity for a non-periodic domain, do the following:

1. Click the **Define...** button under **Periodic Repeats** in the *Views Dialog Box* (p. 2000).

   Figure 31.55 The Graphics Periodicity Dialog Box



2. In the resulting *Graphics Periodicity Dialog Box* (p. 2003) (*Figure 31.55* (p. 1443)), select the **Cell Zone** for which you want to specify the number of repeats.

   Associated Surfaces *list contains the surfaces associated with the selected cell zone. This is only informative and you cannot edit the selection of surfaces in this box.*

3. Specify **Rotational** or **Translational** as the **Periodic Type**.
4. For translational periodicity, specify the **Translation** distance of the repeated domain in the **X**, **Y**, and **Z** directions. For rotational periodicity, specify the axis about which the periodicity is defined and the **Angle** by which the domain is rotated to create the periodic repeat. For 3D problems, the axis of rotation is the vector passing through the specified **Axis Origin** and parallel to the vector from (0,0,0) to the (**X,Y,Z**) point specified under **Axis Direction**. For 2D problems, you will specify only the **Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point.
5. Specify **Number of Repeats** for the selected cell zone.
6. Click **Set** in the *Graphics Periodicity Dialog Box* (p. 2003).
7. Follow the same procedure for other cell zones.
8. Click **Apply** in the *Views Dialog Box* (p. 2000) to visualize the modified display.

You can delete the definition of any periodicity you have defined for graphics by clicking on the **Reset** button in the *Graphics Periodicity Dialog Box* (p. 2003).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

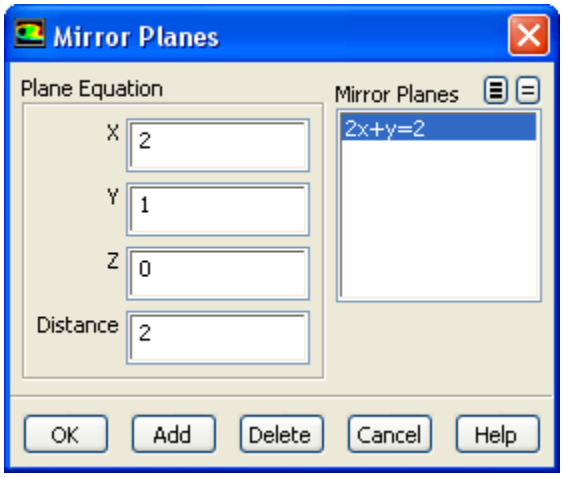
For the 3D domain with multiple periodic zones having different periodicity, ANSYS FLUENT can repeat only mesh, contour and vector plots, and not the pathlines and particle tracks. Also if such domain contains, isosurfaces and clip-surfaces, that are associated with a particular cell zone, they are repeated using the same periodicity that is defined for that cell zone. However, if the surface is not associated with any cell zone, you cannot specify the periodicity for that surface.

31.5.5.2. Mirroring for Graphics

To define a mirror plane for a non-symmetric domain, do the following:

1. Click the **Define Plane...**button under **Mirror Planes** in the *Views Dialog Box* (p. 2000).

   Figure 31.56 The Mirror Planes Dialog Box



2. In the resulting *Mirror Planes Dialog Box* (p. 2002) (*Figure 31.56* (p. 1444)), set the coefficients of **X**, **Y**, and **Z** and the **Distance** (of the plane from the origin) in the following equation for the mirror plane:

$$Ax + By + Cz = \text{distance} \tag{31–1}$$

3. Click the **Add** button to add the defined plane to the **Mirror Planes** list. When you are done creating mirror planes, click **OK**. The newly defined plane(s) will now appear in the **Mirror Planes** list in the *Views Dialog Box* (p. 2000). To include the mirroring in the display, select the plane(s) and click **Apply**, as described above.

If you want to delete a mirror plane that you have defined, select it in the **Mirror Planes** list in the *Mirror Planes Dialog Box* (p. 2002) and click the **Delete** button. When you click **OK** in this dialog box, the deleted plane will be removed permanently from the **Mirror Planes** list in the *Views Dialog Box* (p. 2000).

31.6. Composing a Scene

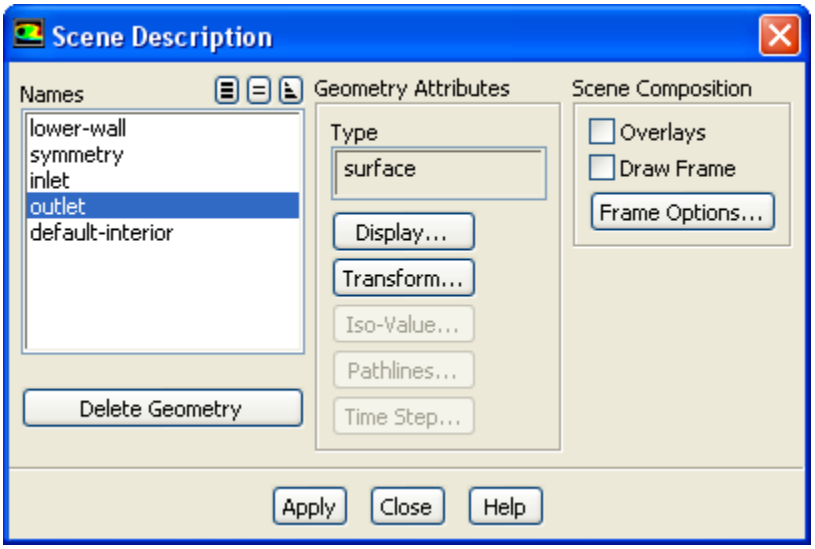
Once you have displayed some geometric objects (meshes, surfaces, contours, vectors, etc.) in your graphics window, you may want to move them around and change their characteristics to increase the effectiveness of the scene displayed. You can use the *Scene Description Dialog Box* (p. 1994) (*Figure 31.57* (p. 1445)) and the *Display Properties Dialog Box* (p. 1996) (*Figure 31.58* (p. 1446)) and *Transformations Dialog Box* (p. 1997) (*Figure*

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.60 (p. 1449)), which are opened from within it, to rotate, translate, and scale each object individually, as well as change the color and visibility of each object.

Graphics and Animations → Scene...

Figure 31.57 The Scene Description Dialog Box



The *Iso-Value Dialog Box* (p. 1998) (*Figure 31.61* (p. 1450)), which is also opened from within the *Scene Description Dialog Box* (p. 1994), allows you to change the isovalue of a selected isosurface. The *Pathline Attributes Dialog Box* (p. 1999) (*Figure 31.62* (p. 1451)) lets you set some pathline attributes. The ability to make geometric objects visible and invisible is especially useful when you are creating an animation (see *Animating Graphics* (p. 1454)) because it allows you to add or delete objects from the scene one at a time. The ability to change the color and position of an object independently of the others in the scene is also useful for setting up animations, as is the ability to change isosurface isovalues. You will find the features in the *Scene Description Dialog Box* (p. 1994) useful even when you are not generating animations because they allow you to manage your graphics window efficiently. The procedure for overlaying graphics, which uses the **Scene Description** dialog box, is described in *Overlay of Graphics* (p. 1411). (Note that you cannot use the *Scene Description Dialog Box* (p. 1994) to control XY plot and histogram displays.)

For additional information, please see the following sections:

31.6.1. Selecting the Object(s) to be Manipulated
31.6.2. Changing an Object's Display Properties
31.6.3. Transforming Geometric Objects in a Scene
31.6.4. Modifying Iso-Values
31.6.5. Modifying Pathline Attributes
31.6.6. Deleting an Object from the Scene
31.6.7. Adding a Bounding Frame

31.6.1. Selecting the Object(s) to be Manipulated

In order to manipulate the objects in the scene, you will begin by selecting the object or objects of interest in the **Names** list in the *Scene Description Dialog Box* (p. 1994) (*Figure 31.57* (p. 1445)). The **Names** list is a list of the geometric objects that currently exist in the scene (including those that are presently invisible). If you select more than one object at a time, any operation (transformation, color specification, etc.) will apply to all the selected objects. You can also select objects by clicking on them in the graphics display using the mouse-probe button, which is, by default, the right mouse button. (See *Controlling the Mouse Button Func-*

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

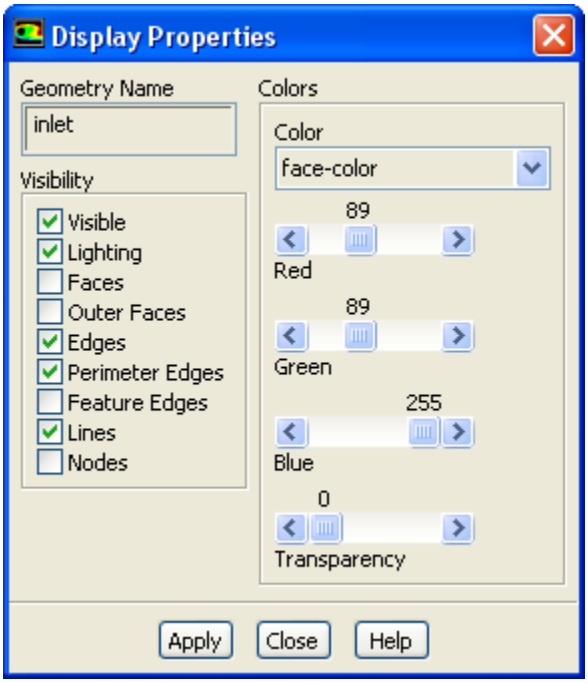
tions (p. 1427) for information about mouse button functions.) To deselect a selected object, simply click on its name in the **Names** list.

When you select one or more objects (either in the **Names** list or in the display), the **Type** field will report the type of the selected object(s). Possible types for a single object include `mesh`, `surface`, `contour`, `vector`, `path`, and `text` (i.e., annotation text). This information is especially helpful when you need to distinguish two or more objects with the same name. When more than one object is selected, the type displayed is `Group`.

31.6.2. Changing an Object's Display Properties

To enhance the scene in the graphics window, you can change the color, visibility, and other display properties of each geometric object in the scene. You can specify different colors for displaying the edges and faces of a mesh object to show the underlying mesh (edges) when the faces of the mesh are filled and shaded. You can also make a selected object temporarily invisible. If, for example, you are displaying the entire mesh for a complicated problem, you can make objects visible or invisible to display only certain boundary zones of the mesh without regenerating the mesh display using the *Mesh Display Dialog Box* (p. 1635). You can also use the visibility controls to manipulate geometric objects for efficient graphics display or for the creation of animations. These features, plus several others, are available in the *Display Properties Dialog Box* (p. 1996) (*Figure 31.58* (p. 1446)).

Figure 31.58 The Display Properties Dialog Box



To set the display properties described above, select one or more objects in the **Names** list in the *Scene Description Dialog Box* (p. 1994) and then click the **Display...** button to open the *Display Properties Dialog Box* (p. 1996) for that object or group of objects.

31.6.2.1. Controlling Visibility

There are several ways for you to control the visibility of an object. All visibility options are listed under the **Visibility** heading in the *Display Properties Dialog Box* (p. 1996).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To make the selected object(s) invisible, turn off the **Visible** option. To "undo" invisibility, enable the **Visible** option again.
- To turn the effect of lighting for the selected object(s) on or off, use the **Lighting** check button. You can choose to have lighting affect only certain objects instead of all of them. Note that if **Lighting** is turned on for an object such as a contour or vector plot, the colors in the plot will not be exactly the same as those in the colormap at the left of the display.
- To toggle the filled display of faces for the selected mesh or surface object(s), use the **Faces** option. Turning **Faces** on here has the same effect as turning it on for the entire mesh in the *Mesh Display Dialog Box* (p. 1635).
- To turn the display of outer edges on or off, use the **Outer Faces** option. This option is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you turn off the **Outer Faces** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (i.e., walls with fluid or solid cells on both sides).
- To turn the display of interior and exterior edges of the geometric object(s) on or off, use the **Edges** option.
- To turn the display of the outline of the geometric object(s) on or off, use the **Perimeter Edges** check button.
- To toggle the display of feature lines (described in *Adding Features to an Outline Display* (p. 1384)), if any, for the selected object(s), use the **Feature Edges** option.
- To toggle the display of the lines (if any) in the geometric object(s), use the **Lines** check button. Pathlines, line contours, and vectors are "lines".
- To toggle the display of nodes (if any) in the geometric object(s), use the **Nodes** check button.

Once you have set the appropriate display parameters, click the **Apply** button to update the graphics display.

31.6.2.2. Controlling Object Color and Transparency

The *Display Properties Dialog Box* (p. 1996) also lets you control an object's color and how transparent it is. All color and transparency options are listed under the **Colors** heading.

- To modify the color of faces, edges, or lines in the selected object(s), choose **face-color**, **edge-color**, **line-color**, or **node-color** in the **Color**drop-down list. The **Red**, **Green**, and **Blue** color scales will show the RGB components of the face, edge or line color, which you can modify by moving the sliders on the color scales. When you are satisfied with the color specification, click **Apply** to save it and update the display. The ability to set the colors for faces and edges can be useful when you wish to have a filled display for the mesh or surface, but you also want to be able to see the mesh lines. You can achieve this effect by specifying different colors for the faces and the edges.
- To set the relative transparency of an object, select **face-color** in the **Color** drop-down list. Move the slider on the **Transparency** scale and click the **Apply** button to update the graphics display. An object with a transparency of 0 is opaque, and an object with a transparency of 100 is transparent. By specifying a high transparency value for the walls of a pipe, for example, you will be able to see contours that you have displayed on cross-sections inside the pipe. This feature is available on all platforms when the software z buffer is used for hidden surface removal, but if your display hardware supports transparency, it will be more efficient to use the hardware z buffer as the hidden surface method instead. You can select these methods in the *Display Options Dialog Box* (p. 1992), as described in *Modifying the Rendering Options* (p. 1425).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

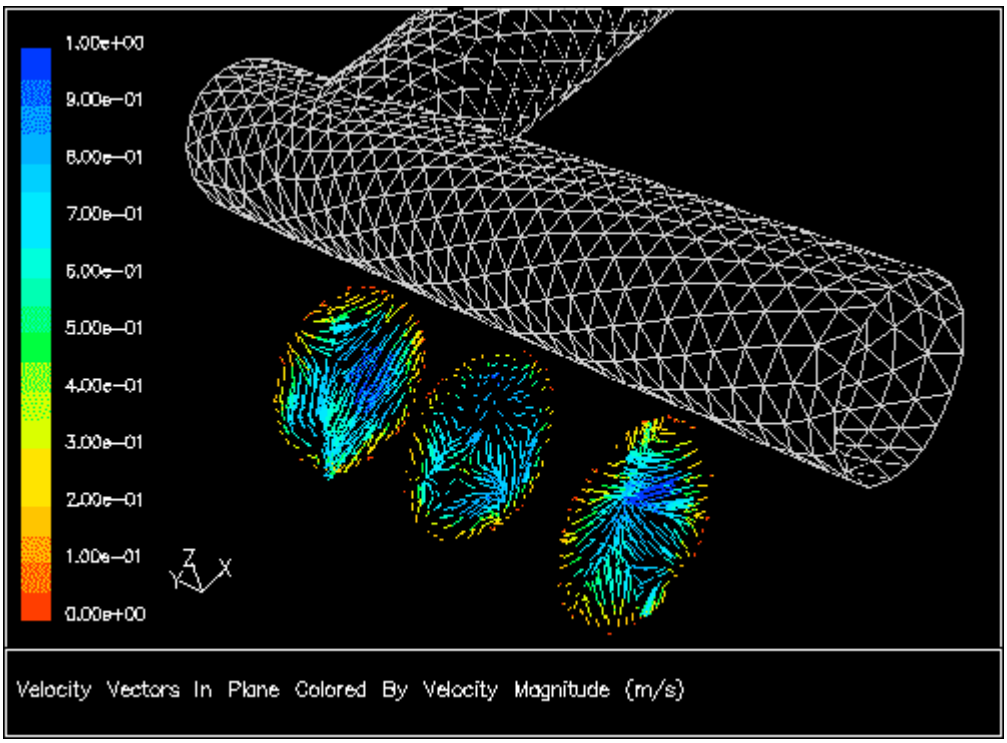
Important

If you save a picture of a display with transparent surfaces, you should *not* set the **File Type** in the *Save Picture Dialog Box* (p. 1988) to **Vector**.

31.6.3. Transforming Geometric Objects in a Scene

When you are composing a scene in your graphics window, you might find it helpful to move a particular object from its original position or to increase or decrease its size. For example, if you have displayed contours or vectors on cross-sections of an internal flow domain (such as a pipe), you might want to translate these cross-sections so that they will appear outside of the pipe, where they can be seen and interpreted more easily. *Figure 31.59* (p. 1448) shows such an example.

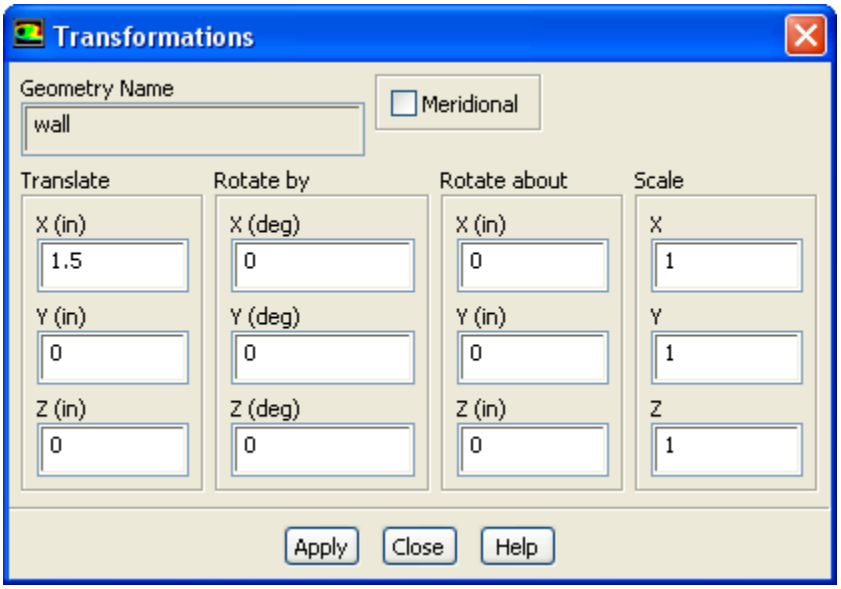
Figure 31.59 Velocity Vectors Translated Outside the Domain for Better Viewing



You can also move an object by rotating it about the x, y, or z axis. If you want to display one object more prominently than the others, you can scale its size. If your geometry is rotating or has rotational symmetry, you can display the meridional view. All of these capabilities are available in the *Transformations Dialog Box* (p. 1997) (*Figure 31.60* (p. 1449)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.60 The Transformations Dialog Box



To perform the transformations described above, select one or more objects in the **Names** list in the *Scene Description Dialog Box* (p. 1994) and then click the **Transform...** button to open the *Transformations Dialog Box* (p. 1997) for that object or group of objects.

31.6.3.1. Translating Objects

To translate the selected object(s), enter the translation distance in each direction in the **X**, **Y**, and **Z** real number fields under **Translate**. Note that you can check the domain extents in the *Scale Mesh Dialog Box* (p. 1634) or the *Iso-Surface Dialog Box* (p. 1933). Translations are not cumulative, so you can easily return to a known state. To return to the original position, simply enter 0 in all three real number fields.

31.6.3.2. Rotating Objects

To rotate the selected object(s), enter the number of degrees by which to rotate about each axis in the **X**, **Y**, and **Z** integer number fields under **Rotate By**. You can enter any value between -360 and 360. By default, the rotation origin will be (0,0,0). If you want to spin an object about its own origin, or about some other point, specify the **X**, **Y**, and **Z** coordinates of that point under **Rotate About**.

Rotations are not cumulative, so you can easily return to a known state. To return to the original position, simply enter 0 in all three integer number fields under **Rotate By**.

31.6.3.3. Scaling Objects

To scale the selected object(s), enter the amount by which to scale in each direction in the **X**, **Y**, and **Z** real number fields under **Scale**. To avoid distortion of the object's shape, be sure to specify the same value for all three entries. Scaling is not cumulative, so you can easily return to a known state. To return the object to its original size, simply enter 1 in all three real number fields.

31.6.3.4. Displaying the Meridional View

To display the meridional view of the selected object(s), enable the **Meridional** option. This option is available only for 3D models. It is applicable to cases with a defined axis of rotation and is especially useful in turbomachinery applications.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The meridional transformation projects the selected entities onto a surface of constant angular coordinate, θ. The resultant projection thus lies in an (r, ζ) plane where ζ is in the direction of the rotation axis and r is normal to it. The value of θ used for the projection is taken as that corresponding to the minimum (r, ζ) point of the entity.

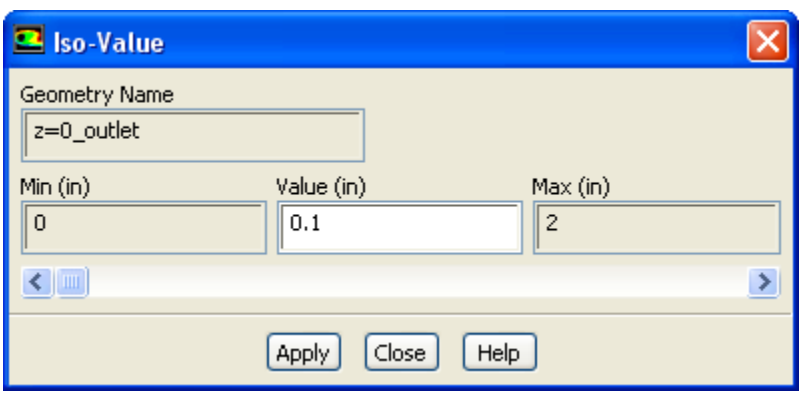
31.6.4. Modifying Iso-Values

One convenient feature that you can use to generate effective animations is the ability to generate surfaces with intermediate values between two isosurfaces with different isovalues. If the surfaces have contours, vectors, or pathlines displayed on them, ANSYS FLUENT will generate and display contours, vectors, or pathlines on the intermediate surfaces that it creates.

31.6.4.1. Steps for Modifying Iso-Values

You can modify an isosurface's isovalue directly by selecting it in the *Scene Description Dialog Box* (p. 1994)'s **Names** list or indirectly by selecting an object displayed on the isosurface. Then click the **Iso-Value...** button to open the *Iso-Value Dialog Box* (p. 1998) (*Figure 31.61* (p. 1450)) for the selected object. Note that this button is available only if the geometric object selected in the **Names** list is an isosurface or an object on an isosurface (contour on an isosurface, for example); otherwise it is grayed out.

Figure 31.61 The Iso-Value Dialog Box



In the *Iso-Value Dialog Box* (p. 1998), set the new isovalue in the **Value** field, and click **Apply**. Contours, vectors, or pathlines that were displayed on the original isosurface will be displayed for the new isovalue.

31.6.4.2. An Example of Iso-Value Modification for an Animation

The ability to generate intermediate surfaces with data displayed on them is especially convenient if you want to create an animation that shows data on successive slices of the problem domain. For example, if you have solved the flow through a pipe junction and you want to create an animation that moves through one of the pipes (along the y axis) and displays pressure contours on several cross-sections, you can use the following procedure:

1. Generate a surface of constant y coordinate (such as the y coordinate at the pipe inlet) using the *Iso-Surface Dialog Box* (p. 1933).
2. Use the *Contours Dialog Box* (p. 1967) to generate contours of static pressure on this isosurface and manipulate the graphics display to the desired view.
3. Open the *Animate Dialog Box* (p. 1986) and create key frame 1.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the *Scene Description Dialog Box* (p. 1994), select the contour in the **Names** list and click the **Iso-Value...** button to open the *Iso-Value Dialog Box* (p. 1998).
5. Change the value of the isovalue to the y coordinate at the other end of the pipe, and click **Apply**. You will see the contours of static pressure at the new y coordinate.
6. Set key frame 10 in the *Animate Dialog Box* (p. 1986).
7. Play back the animation.

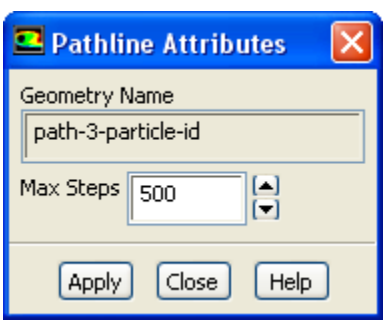
When you play back the animation, ANSYS FLUENT will create the intermediate frames showing contours of static pressure on the slices between the two ends of the pipe. Ten slices will be shown in succession, all with contours displayed on them.

Using the *Sweep Surface Dialog Box* (p. 1984) to animate the display of contours or vectors on a surface that sweeps through the domain may be more convenient than the procedure described above. See *Displaying Results on a Sweep Surface* (p. 1409) for details.

31.6.5. Modifying Pathline Attributes

If you are creating animations of existing pathlines, you may want to change the number of steps used in the computation of the pathlines. This allows you to animate pathlines advancing through the domain. To do so, select the pathlines in the **Names** list in the *Scene Description Dialog Box* (p. 1994) and then click the **Pathlines...** button to open the *Pathline Attributes Dialog Box* (p. 1999) (*Figure 31.62* (p. 1451)).

Figure 31.62 The Pathline Attributes Dialog Box



In the *Pathline Attributes Dialog Box* (p. 1999), set the new maximum number of steps for pathline computation (**Max Steps**). After you change the value and click **Apply**, the selected pathline will be recomputed and re-drawn.

31.6.5.1. An Example of Pathline Modification for an Animation

You can use the following procedure to animate pathlines from step 2 to step 101 (for example):

1. Generate the plot of pathlines using the *Pathlines Dialog Box* (p. 1975).
2. In the *Scene Description Dialog Box* (p. 1994), select the pathlines in the **Names** list and click the **Pathlines...** button to open the *Pathline Attributes Dialog Box* (p. 1999).
3. Change the value of the maximum number of steps to 2, and click **Apply**.
4. Open the *Animate Dialog Box* (p. 1986) and create key frame 1.
5. In the *Pathline Attributes Dialog Box* (p. 1999), change the value of the maximum number of steps to 101, and click **Apply**.
6. Set key frame 100 in the *Animate Dialog Box* (p. 1986).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Play back the animation.

When you play back the animation, ANSYS FLUENT will animate the pathlines so that they advance one step in each frame.

31.6.6. Deleting an Object from the Scene

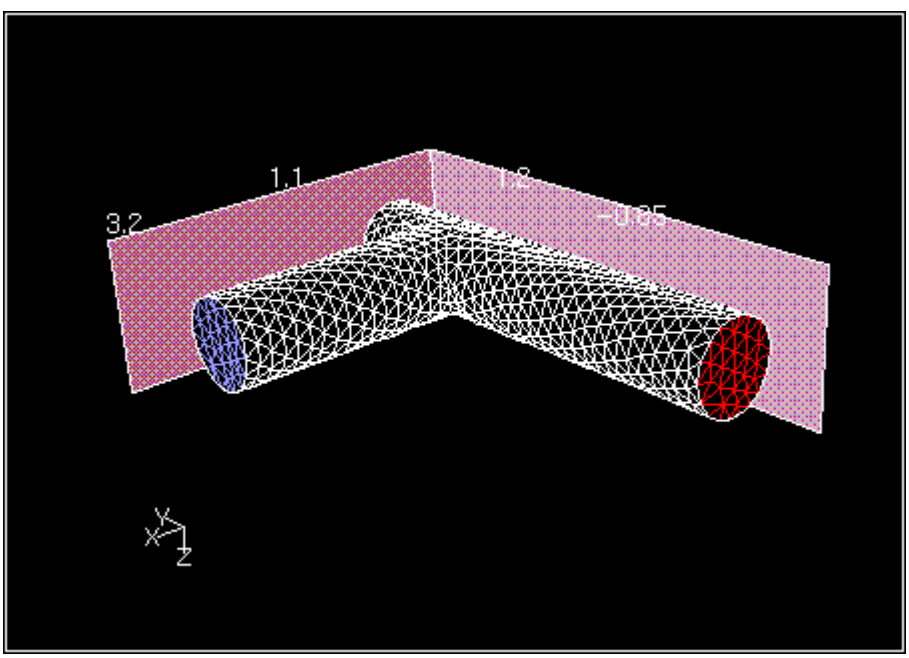
If you are composing a complex scene with overlays and find that you no longer want to keep one of the objects, it is possible to delete it without affecting any of the other objects in the scene. The ability to delete individual objects is especially useful if you have overlays on and you generate an unwanted object (e.g., if you generate contours of the wrong variable). You can simply delete the unwanted object and continue your scene composition, instead of starting over from the beginning. Note that it is also possible to make objects temporarily invisible, as described in *Controlling Visibility* (p. 1446).

Object deletion is performed in the *Scene Description Dialog Box* (p. 1994) (*Figure 31.57* (p. 1445)). To delete an object from the scene, select it in the **Names** list and then click the **Delete Geometry** button. The selected name will disappear from the **Names** list, and the display will be updated immediately.

31.6.7. Adding a Bounding Frame

ANSYS FLUENT allows you to add a bounding frame around your displayed domain. You may also include measure markings on the bounding frame to indicate the length, height, and/or width of the domain, as shown in *Figure 31.63* (p. 1452).

Figure 31.63 Graphics Display with Bounding Frame

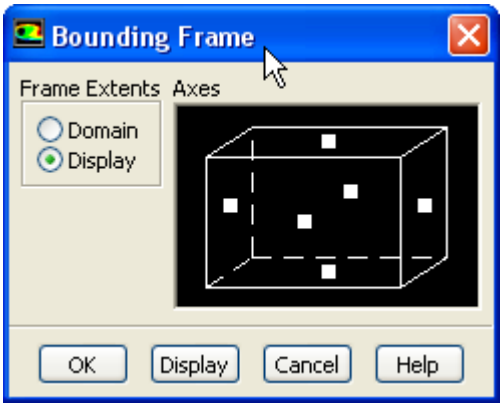


To add a bounding frame to your display, you will follow the procedure below:

1. Click the **Frame Options...** button in the *Scene Description Dialog Box* (p. 1994) (*Figure 31.57* (p. 1445)) to open the *Bounding Frame Dialog Box* (p. 2000) (*Figure 31.64* (p. 1453)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.64 The Bounding Frame Dialog Box



2. Under **Frame Extents** in the *Bounding Frame Dialog Box* (p. 2000), select **Domain** or **Display** to indicate whether the bounding frame should encompass the domain extents or only the portion of the domain that is shown in the display.
3. In the **Axes** portion of the *Bounding Frame Dialog Box* (p. 2000), specify the frame boundaries and measurements to be shown in the display:
   - Indicate the bounding plane(s) (e.g., the x- z and y- z planes shown in *Figure 31.63* (p. 1452)) to be displayed by clicking on the white square on the appropriate plane of the box shown under the **Axes** heading. You can use any of the mouse buttons. The square will turn red to indicate that the associated bounding plane will be displayed in the graphics window.
   - Specify where you would like to see the measurement annotations by clicking on the appropriate edge of the box. The edge will turn red to indicate that the markings will be displayed along that edge of the displayed geometry.

   Important

   If you have trouble determining which square or edge corresponds to which location in your domain, you can easily find out by displaying one or two bounding planes to get your bearings. You can then select the appropriate objects to obtain the final display.

4. Click the **Display** button to update the display with the current settings. If you are not satisfied with the frame, repeat steps 2 and/or 3 and click **Display** again.
5. Once you are satisfied with the bounding frame that is displayed, click **OK** to close the *Bounding Frame Dialog Box* (p. 2000) and save the frame settings for future displays.
6. If you wish to include the bounding frame in all subsequent displays, enable the **Draw Frame** option in the *Scene Description Dialog Box* (p. 1994) and click **Apply**. If this option is not enabled, the bounding box will appear only in the current display; it will not be redisplayed when you generate a new display (unless you have overlays enabled).

The bounding planes and axis annotations will appear in the **Names** list of the *Scene Description Dialog Box* (p. 1994), and you can manipulate them in the same way as any other geometric object in the display. For example, you can use the *Display Properties Dialog Box* (p. 1996) to change the face color of a bounding plane or to make it transparent (see *Changing an Object's Display Properties* (p. 1446)).

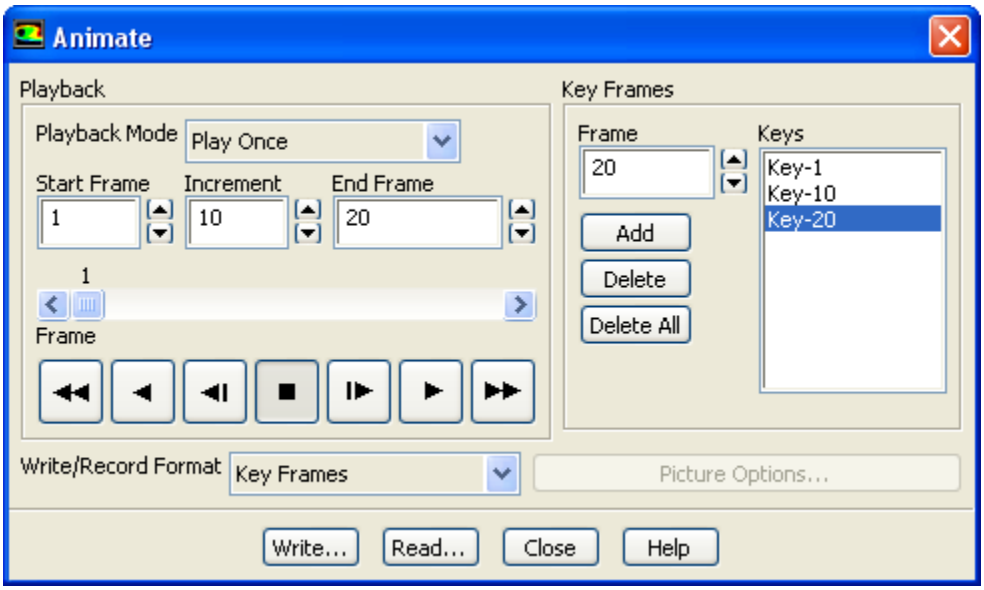
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.7. Animating Graphics

To generate animations that progress from one static view of the graphics display to the next, you can set up "key frames" (individual static images) using the *Animate Dialog Box* (p. 1986) (*Figure 31.65* (p. 1454)).

Graphics and Animations → **Scene Animation**

Figure 31.65 The Animate Dialog Box



You can compose a scene in the graphics window and define it as a single key frame. Then, modify the scene by moving or scaling objects, making some objects invisible or visible, changing colors, changing the view, or making other changes, and define the new scene as another key frame. ANSYS FLUENT can then interpolate smoothly between the two frames that you defined, creating a specified number of intermediate frames.

Another method of generating animations is to automatically generate surfaces with intermediate values between two isosurfaces with different isovalues. See *Modifying Iso-Values* (p. 1450) for details. See *Displaying Results on a Sweep Surface* (p. 1409) for information about displaying the mesh, contours, or vectors on a surface that sweeps through the domain. If you want to create a graphical animation of the solution over time, you can use the *Solution Animation Dialog Box* (p. 1953) to set up the graphical displays that you want to use in the animation. You can choose the type of display you want to animate by choosing **Mesh**, **Contours**, **Pathlines**, **Particle Tracks**, **Vectors**, **XY Plot**, or **Monitor**. For details on animating the solution, see *Animating the Solution* (p. 1296). For more information on generating, displaying, and saving pathlines and particle tracks, refer to *Displaying Pathlines* (p. 1400).

For additional information, please see the following sections:

31.7.1. Creating an Animation
31.7.2. Playing an Animation
31.7.3. Saving an Animation
31.7.4. Reading an Animation File
31.7.5. Notes on Animation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.7.1. Creating an Animation

You can define any number of key frames (up to 3000) to create your animation. By assigning the appropriate numbers to the key frames, you provide the information ANSYS FLUENT needs to create the correct number of intermediate frames. For example, to create a simple animation that begins with a front view of an object, moves to a side view, and ends with a rear view of the object, you would follow the procedure outlined below:

1. Determine the number of frames that you want in the animation. For this example, consider the animation to be 31 frames.
2. Determine the number of key frames that you need to specify. In this example, you will specify three: one showing the front view, one showing the side view, and one showing the rear view.
3. Determine the appropriate key frame numbers to assign to the 3 specified frames. Here, the front view will be specified as key frame 1, the side view will be key frame 16, and the rear view will be key frame 31.
4. Compose the scenes for each view to be used as a key frame. You can use the *Scene Description Dialog Box* (p. 1994) (see *Composing a Scene* (p. 1444)) and the *Views Dialog Box* (p. 2000) (see *Modifying the View* (p. 1433)) to modify the display, and any other dialog boxes or commands to create contours, vectors, pathlines, etc. to be included in each scene. After you complete each scene, create the appropriate key frame by setting the **Frame** number and clicking on the **Add** button under **Key Frames** in the *Animate Dialog Box* (p. 1986). For special considerations related to key frame definition, see *Notes on Animation* (p. 1458).

   Important

   Be sure to change the **Frame** number before clicking on the **Add** button, or you will overwrite the last key frame that you created.

   You can check any of the key frames that you have created by selecting it in the **Keys** list. The selected key frame will be displayed in the graphics window.
5. When you complete the animation, you can play it back as described in *Playing an Animation* (p. 1455) and/or save it as described in *Saving an Animation* (p. 1456).

31.7.1.1. Deleting Key Frames

If, during the creation of your animation, you want to remove one of the key frames that you have defined, select the key frame in the **Keys** list and click the **Delete** button. If you want to delete all key frames and start over again, click the **Delete All** button.

31.7.2. Playing an Animation

Once you have defined the key frames (as described in *Creating an Animation* (p. 1455)) or read in a previously created animation file (as described in *Reading an Animation File* (p. 1458)), you can play back the animation and ANSYS FLUENT will interpolate between the frames that you specified to complete the animation.

To play the animation once through from start to finish, click the "play" button under the **Playback** heading in the *Animate Dialog Box* (p. 1986). (The buttons function in a way similar to those on a standard video cassette player. "Play" is the second button from the right—a single triangle pointing to the right.) To play the animation backwards once, click the "play reverse" button (the second from the left—a single triangle point to the left). As the animation plays, the **Frame** scale shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If, instead of playing the complete animation, you

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

want to jump to a particular key frame, move the **Frame** slider bar to the desired frame number, and the frame corresponding to the new frame number will be displayed in the graphics window.

Additional options for playing back animations are described below. Be sure to check *Notes on Animation* (p. 1458) as well for important notes about playing back animations.

31.7.2.1. Playing Back an Excerpt

You may sometimes want to play only one portion of a long animation. To do this, you can modify the **Start Frame** and the **End Frame** under the **Playback** heading in the *Animate Dialog Box* (p. 1986). For example, if your animation contains 50 frames, but you want to play only frames 20 to 35, you can set **Start Frame** to 20 and **End Frame** to 35. When you play the animation, it will start at frame 20 and finish at frame 35.

31.7.2.2. "Fast-Forwarding" the Animation

You can "fast-forward" or "fast-reverse" the animation by skipping some of the frames during playback. To fast-forward the animation, set the **Increment** and click the fast-forward button. If, for example, your **Start Frame** is 1, your **End Frame** is 15, and your **Increment** is 2, when you click the fast-forward button (the last button on the right—two triangles pointing to the right), the animation will show frames 1, 3, 5, 7, 9, 11, 13 and 15. Clicking on the fast-reverse button (the first button on the left—two triangles pointing to the left) will show frames 15, 13, 11,...1.

31.7.2.3. Continuous Animation

If you want the playback of the animation to repeat continuously, there are two options available.

- To continuously play the animation from beginning to end (or from end to beginning, if you use one of the reverse play buttons), select **Auto Repeat** in the **Playback Mode** drop-down list in the *Animate Dialog Box* (p. 1986).
- To play the animation back and forth continuously, reversing the playback direction each time, select **Auto Reverse** in the **Playback Mode** drop-down list.

To turn off the continuous playback, select **Play Once** in the **Playback Mode** list. This is the default setting.

31.7.2.4. Stopping the Animation

To stop the animation during playback, click the "stop" button (the square in the middle of the playback control buttons). If your animation contains very complicated scenes, there may be a slight delay before the animation stops.

31.7.2.5. Advancing the Animation Frame by Frame

To advance the animation manually frame by frame, use the third button from the right (a vertical bar with a triangle pointing to the right). Each time you click this button, the next frame will be displayed in the graphics window. To reverse the animation frame by frame, use the third button from the left (a left-pointing triangle with a vertical bar). Frame-by-frame playback allows you to freeze the animation at points that are of particular interest.

31.7.3. Saving an Animation

Once you have created your animation, you can save it in any of the following formats:

- Animation file containing the key frame descriptions

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Picture files, each containing a frame of the animation
- MPEG file containing each frame of the animation
- Video (see *Creating Videos* (p. 1458))

31.7.3.1. Animation File

You can save the key frame definitions to a file that can be read back into ANSYS FLUENT (see *Reading an Animation File* (p. 1458)) when you want to replay the animation. Since the animation file will contain only the key frame definitions, you must be sure that you have a case and data file containing the necessary surfaces and other information referred to by the key frame descriptions.

To write an animation file, select **Key Frames** in the **Write/Record Format** drop-down list in the *Animate Dialog Box* (p. 1986), and click the **Write...** button. In *The **Select File** Dialog Box* (p. 33), specify the name of the file and save it.

31.7.3.2. Picture File

You can also generate a picture file for each frame in the animation. This feature allows you to save your animation frames to picture files used by an external animation program such as ImageMagick. To save the animation as a picture file, follow these steps:

1. Select **Picture Files** in the **Write/Record Format** drop-down list in the *Animate Dialog Box* (p. 1986).
2. If necessary, click the **Picture Options...** button to open the *Save Picture Dialog Box* (p. 1988) and set the appropriate parameters for saving the picture files. (If you are saving picture files for use with ImageMagick, for example, you may want to select the window dump format. See *Window Dumps (Linux/UNIX Systems Only)* (p. 121) for details.) Click **Apply** in the *Save Picture Dialog Box* (p. 1988) to save your modified settings.

   Important

   Do not click the **Save...** button in the *Save Picture Dialog Box* (p. 1988). You will save the picture files from the *Animate Dialog Box* (p. 1986) in the next step.

3. In the *Animate Dialog Box* (p. 1986), click the **Write...** button. In *The **Select File** Dialog Box* (p. 33), specify the filename and click **OK** to save the files. (See *Window Dumps (Linux/UNIX Systems Only)* (p. 121) for information about specifying filenames that increment automatically as additional pictures are saved.). ANSYS FLUENT will replay the animation, saving each frame to a separate file.

31.7.3.3. MPEG File

It is also possible to save all of the frames of the animation in an MPEG file, which can be viewed using an MPEG decoder such as **mpeg_play**. Saving the entire animation to an MPEG file will require less disk space than storing the individual window dump files (using the picture method), but the MPEG file will yield lower-quality images. To save the animation to an MPEG file, follow these steps:

1. Select **MPEG**in the **Write/Record Format** drop-down list in the *Animate Dialog Box* (p. 1986).
2. In the *Animate Dialog Box* (p. 1986), click the **Write...** button. In *The **Select File** Dialog Box* (p. 33), specify the filename and click **OK** to save the files.

   ANSYS FLUENT will replay the animation and save each frame to a separate scratch file, and then it will combine all the files into a single MPEG file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.7.4. Reading an Animation File

If you have saved the key frames defining an animation to an animation file (as described in *Saving an Animation* (p. 1456)), you can read that file back in at a later time (or in different session) and play the animation. Before reading in an animation file, be sure that the current case and data contain the surfaces and any other information that the key frame description refers to.

To read an animation file, click the **Read...** button in the *Animate Dialog Box* (p. 1986). In *The **Select File** Dialog Box* (p. 33), specify the name of the file to be read.

31.7.5. Notes on Animation

When you are creating and playing back animations, please note the following:

- For smoother animations, enable the **Double Buffering** option in the *Display Options Dialog Box* (p. 1992) (see *Modifying the Rendering Options* (p. 1425)). This will reduce screen flicker during graphics updates.
- When you are defining key frames, you must create all geometric objects that will be used in the animation before you create any key frames. You cannot create a key frame using one set of geometric objects and then generate a new geometry (such as a vector plot) and include that in another key frame. Create all geometric objects first, and then use the *Display Properties Dialog Box* (p. 1996) to control the visibility of the objects in each key frame (see *Controlling Visibility* (p. 1446)).
- A single animation sequence can contain up to 3000 key frames.
- When you play back an animation, the colormap used will be the one that is currently active, *not* the one that was active during "recording."

31.8. Creating Videos

Tools are available for creating videos from ANSYS FLUENT. This section is a guide to video creation using the new video capabilities. It assumes that you have a ready-to-use video system, and that you are familiar with this system, including the special video hardware and software installed on your computer. The main use for this feature is to record an animation that you have created using the *Animate Dialog Box* (p. 1986) (as described in *Animating Graphics* (p. 1454)). This section will describe issues involved in recording animations to video, the kind of video equipment you will need, and the procedures for creating a video using ANSYS FLUENT.

Important

Video creation is not currently available in Windows versions of ANSYS FLUENT.

For additional information, please see the following sections:

31.8.1. Recording Animations To Video
31.8.2. Equipment Required
31.8.3. Recording an Animation with ANSYS FLUENT

31.8.1. Recording Animations To Video

Recording an animation involves copying the computer-generated images to videotape so that you can view the animation with a VCR, or another type of tape player. This task is not an easy one, as there are several issues that should be addressed in order to create an acceptable video. A couple of these issues are described in the following sections.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.1.1. Computer Image vs. Video Image

The computer monitor uses a different video signal than the video tape recorder (VTR). Most computers use an RGB-component, non-interlaced signal with high resolution and a high refresh rate. A VTR typically uses a standard broadcast video signal (such as NTSC or PAL), which has an interlaced, composite signal with lower resolution and a lower refresh rate. In order to send the computer image to the VTR for recording, the computer has to produce a video signal in the proper format. This requires extra hardware, which, in many cases, converts RGB component video to standard broadcast video, resulting in a lower quality image. A solution to this problem is to make sure that the image you are recording does not have small text, or too much small detail that will be hard to see on video. Sometimes it is best to zoom in on an area of interest in a large image and animate just that portion.

Another problem is that RGB-component video has a larger color space (or color gamut) than standard broadcast video. This means that some colors may get "clipped" when an image is converted to broadcast video, resulting in washed-out colors, or color bleeding. The solution is to try to make sure that the colors fall within the color space of the video format, and are not oversaturated. Some picture controls that can help you do this are available in ANSYS FLUENT. These controls will be discussed in *Check the Picture Quality* (p. 1464).

31.8.1.2. Real-Time vs. Frame-By-Frame

If the images in the animation can be rendered fast enough on the computer screen, it may be possible to record the animation in real-time. This is as simple as placing the video tape recorder (VTR) in record mode, and playing the animation on the computer screen. This also requires scan-converting hardware that will convert the scan lines of the computer screen to a video signal sent to the VTR.

In many cases, however, the animation cannot be played back on the computer screen in real-time. To create a video that plays the animation at a desirable speed, the animation must be recorded frame-by-frame. This involves sending one frame to the VTR, instructing it to record the frame at a specific point on the tape, then backing up the VTR to repeat the procedure with the next frame. This process takes quite a bit longer than real-time recording, but the result can be a much smoother video animation.

31.8.2. Equipment Required

In general, recording an animation to video requires a system with the following hardware components:

Computer
with video hardware to produce the video signal.

Editing VTR
(video tape recorder) that supports frame-accurate recording.

VTR Controller
which enables computer software to control the recording process.

Two VTR controller models are supported by ANSYS FLUENT: the V-LAN controller developed by Videomedia, Inc., and the MiniVAS/MiniVAS-2 controller developed by the V.A.S. Group. ANSYS FLUENT assumes that your recording system is set up as shown in *Figure 31.66* (p. 1460) for a system with a V-LAN controller or as shown in *Figure 31.67* (p. 1460) for a system with a MiniVAS controller.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.66 Recording System with V-LAN Controller

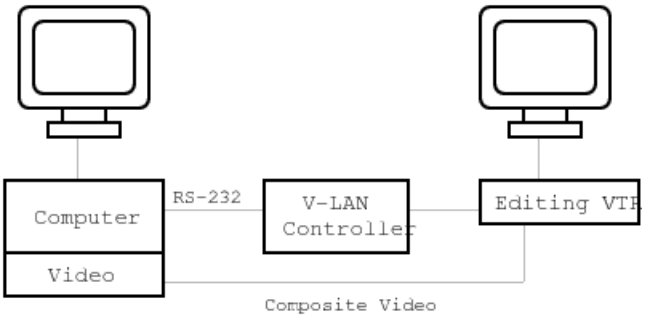
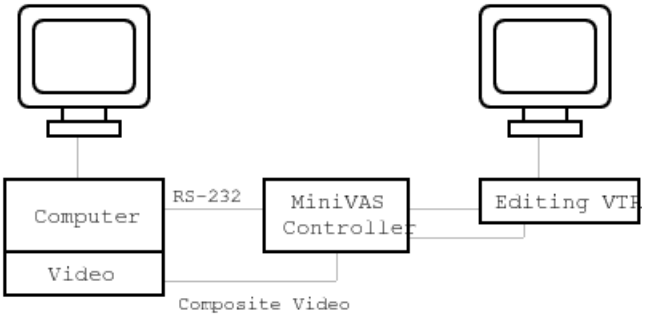


Figure 31.67 Recording System with MiniVAS/MiniVAS-2 Controller



31.8.3. Recording an Animation with ANSYS FLUENT

The procedure for recording an animation using ANSYS FLUENT is as follows:

- *Create an Animation* (p. 1460)
- *Open a Connection to the VTR Controller* (p. 1461)
- *Set Up Your Recording Session* (p. 1461)
- *Check the Picture Quality* (p. 1464)
- *Make Sure Your Tape is Formatted (Preblacked)* (p. 1465)
- *Start the Recording Session* (p. 1465)

Each step is described in detail in the following sections.

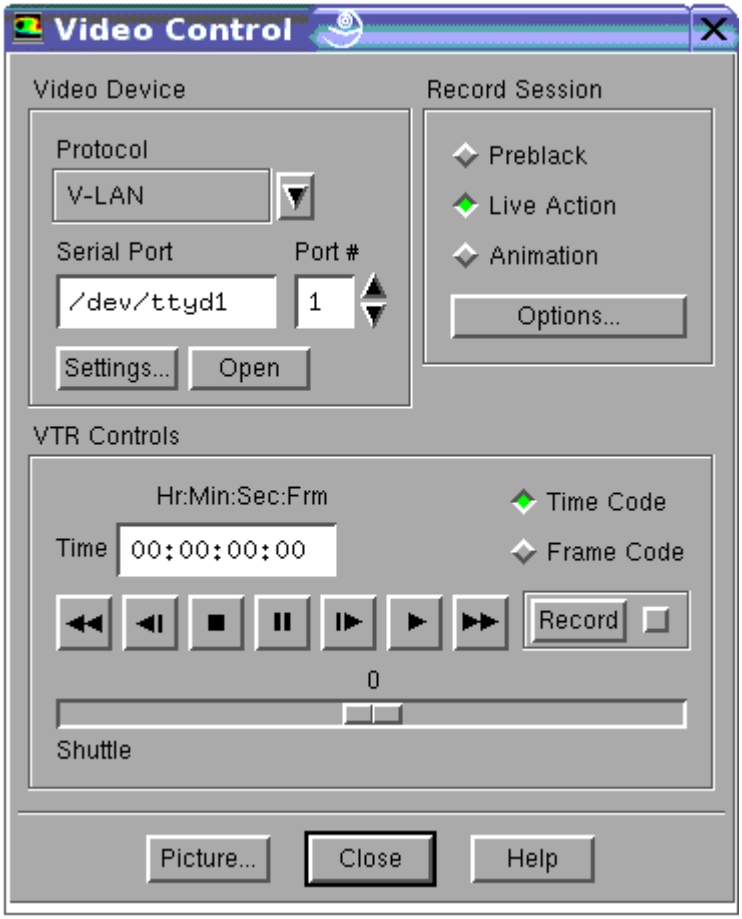
31.8.3.1. Create an Animation

When recording animations to video, you must first create your animation. It's also a good idea to play it back a couple times to make sure you are satisfied with it, and to save the animation key frame definitions to a file for later use (see *Creating an Animation* (p. 1455)).

When you are ready to record the animation, you can select **Video** in the **Write/Record Format** drop-down list found in the *Animate Dialog Box* (p. 1986). When you do so, the name of the **Write...** button will change to **Record...**, and you can click **Record...** to display the *Video Control Dialog Box* (p. 2148) (*Figure 31.68* (p. 1461)) used for video creation. This dialog box can also be displayed by selecting the **Video Control...** menu item in the **Display** pull-down menu.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.68 The Video Control Dialog Box



31.8.3.2. Open a Connection to the VTR Controller

The procedure for connecting to your VTR controller is as follows:

1. Select the protocol used by your VTR controller using the **Protocol** drop-down list.
2. Check the settings for your VTR controller by clicking on the **Settings...** button. For V-LAN, this will display the *V-LAN Settings Dialog Box* (p. 2150), and for MiniVAS, it will display the *MiniVAS Settings Dialog Box* (p. 2152).
3. Select the RS-232 serial port used to connect the VTR controller to your computer. Usually, the serial port is identified by a file name such as `/dev/ttyd1` for serial port 1, and `/dev/ttyd2` for serial port 2. If this is the case on your system, you can simply set the value of **Port #**; otherwise, you can type a new file name in the **Serial Port** text entry. Make sure that you have the proper UNIX read/write permissions for the file.
4. Open a connection to the VTR controller by clicking the **Open** button. If successful, a line will be printed out in the console window that reports the VTR controller protocol version and the VTR device ID.

31.8.3.3. Set Up Your Recording Session

Once you have established a connection to the VTR controller, you can set up your recording session. There are three types of recording sessions, as described below:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Preblack

is the process of formatting a tape by laying down a time code onto the tape. A tape must be formatted before any frame-accurate editing, including frame-by-frame animation, can be performed. During this process, one usually records a black video signal onto the tape as well, thus the name "preblack". When you select this option, the current graphics window will be cleared to black. You can use the window to send your black video signal to the VTR.

Important

When you preblack a previously formatted tape, a new time code will be written and any previously recorded video will be destroyed.

Live Action

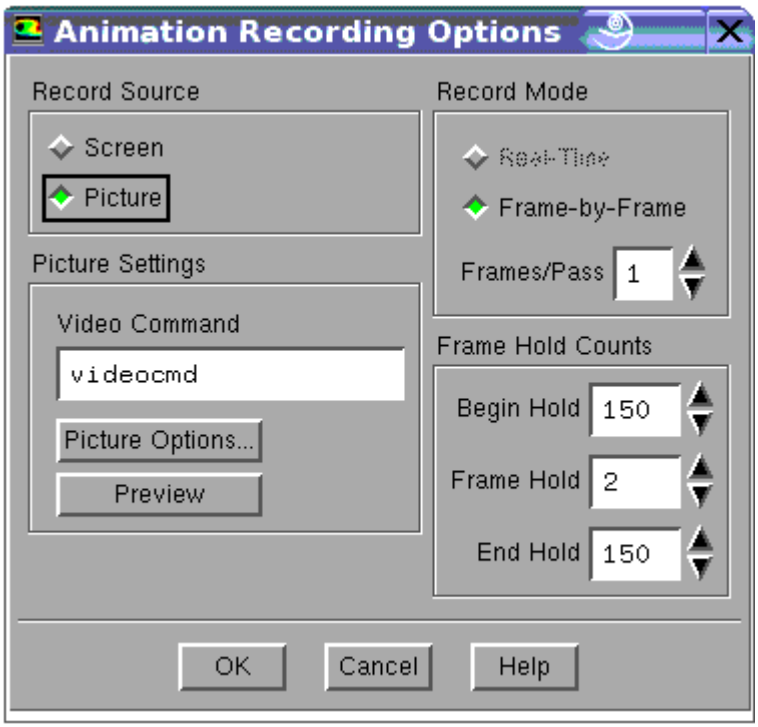
allows you to record a live ANSYS FLUENT session which can be used for demonstration. This option requires your computer's video hardware to have a scan converter that will send the computer display image to your VTR system.

Animation

will play an animation that you have created, and record it onto your VTR system.

The **Options...** button in the *Video Control Dialog Box* (p. 2148) is used to display the *Animation Recording Options Dialog Box* (p. 2153) (*Figure 31.69* (p. 1462)):

Figure 31.69 The Animation Recording Options Dialog Box



There are three parts to setting up your animation recording session:

1. Select the recording source.
2. Choose real-time or frame-by-frame recording.
3. Set the video frame hold counts.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.3.3.1. Select the Recording Source

There are two possible video sources that can be used for recording an animation: **Screen** and **Picture**. The choice of video source depends on what your video hardware/software provides. Here is a description of each:

Screen
can be used if your computer's video hardware can send all or a portion of the computer screen as a video signal to the VTR using a scan converter and associated software. With this option, you are responsible for setting up the scan converter and sending the video signal to the VTR.

Picture
instructs ANSYS FLUENT to create a picture of each frame of animation and send the picture file to the computer's video hardware using a system command. This option assumes that your computer's video system includes a frame buffer that can store an image and send it as a video signal to the video recording system.

When using the picture option, a shell script will be called that will send the picture file to the video frame buffer. The default setting is `videocmd`, which is a shell script that is included in your ANSYS FLUENT distribution. It is located in *path* `/Fluent.Inc/bin`, where *path* is the folder in which you have placed the release folder, `Fluent.Inc`. This shell script will execute your system's command to send an image file to the video frame buffer. The script `videocmd` is set up to call the SGI system command `memtovid`. If you have a different system, you must copy the shell script `videocmd` to a new file and modify it to perform the proper task on your system (see the comments in `videocmd` for details). You can specify the name of your shell script using the **Video Command** text entry in the *Animation Recording Options Dialog Box* (p. 2153).

In order to send a picture file of the proper format to the video frame buffer, you must set up the picture format using the *Save Picture Dialog Box* (p. 1988), which can be displayed by clicking the **Picture Options...** button in the *Animation Recording Options Dialog Box* (p. 2153). If you choose to perform a window dump to create the picture file, the default window dump command used will also be `videocmd`. You can change this setting to use your own command. After setting the picture options, click **Apply** instead of **Save...** in the *Save Picture Dialog Box* (p. 1988) to apply the change.

Once you have set up the picture format and system command, you can test the configuration by sending the picture in the current graphics window to the video frame buffer. This is done by clicking on **Preview** in the *Animation Recording Options Dialog Box* (p. 2153). Note that this is another way to send a black video signal to your VTR when you are preblacking a tape.

31.8.3.3.2. Choose Real-Time or Frame-By-Frame Recording

There are two methods for recording an animation: real-time and frame-by-frame.

Real-Time
can be used if the animation playback speed is fast enough to provide a reasonably smooth animation in real-time. This is only available if the selected record source is **Screen**. In this mode, ANSYS FLUENT will simply turn VTR recording on, play the animation, then stop the recording.

Frame-By-Frame
is used to produce a higher-quality video animation by recording one frame at a time. For each animation frame, this method will 1) play the frame on the screen (and generate the picture file, if needed), 2) preroll the VTR, and 3) record the frame. If the animation has 50 frames, this procedure is repeated 50 times, i.e., 50 record passes are made. This is the recommended method, because the real-time playback of the animation will usually be too slow and choppy.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When recording in frame-by-frame mode, there is an optional setting called **Frames/Pass**, which can be used to try and speed up the frame-by-frame recording process. It specifies the number of animation frames recorded to tape per record pass. If the animation is long enough (200 frames or more), you can try setting this value to 2 or higher. For example, if you set this value to 2 for a 202-frame animation, it will record animation frame 1 during the first pass, frames 2 and 102 during the second pass, frames 3 and 103 during the third pass, and so on. This is possible only if the animation frames can be rendered in time to be inserted onto the tape during a record pass, so use this setting with caution.

31.8.3.3.3. Set the Video Frame Hold Counts

The video standard NTSC has a frame rate of 30 frames/sec (and the PAL standard has a rate of 25 frames/sec). At the NTSC rate, a 150-frame animation will take only 5 seconds to play. To stretch out the animation, you can record the same animation frame over 2 or more video frames. This is done by setting video frame hold counts for the beginning, middle, and end of the animation, using the *Animation Recording Options Dialog Box* (p. 2153) controls.

Begin Hold
: specifies the number of video frames to hold the first animation frame. It helps to hold the first frame for about 5 seconds (150 video frames for NTSC, or 125 for PAL) so that the viewer can get accustomed to the picture before the animation begins.

Frame Hold
: specifies the number of video frames to hold each animation frame, other than the first and last. To slow down your recorded animation, try setting this value to 2 or 3.

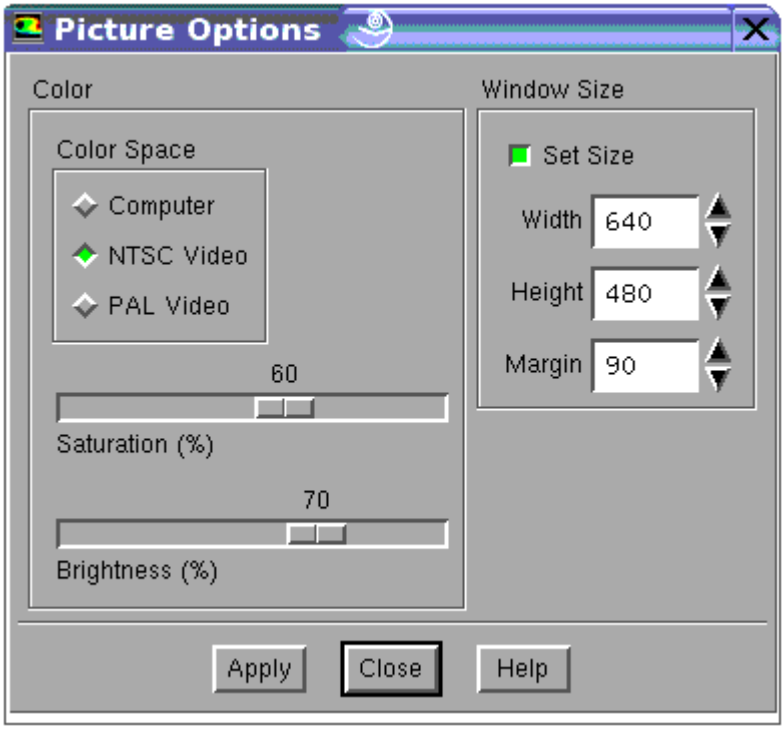
End Hold
: specifies the number of video frames to hold the last animation frame. You may want to hold the last animation frame for about 5 seconds to provide closure.

31.8.3.4. Check the Picture Quality

As described in *Recording Animations To Video* (p. 1458), there are several sacrifices made when sending a computer image to video, including loss of color and resolution. Some steps can be taken to minimize the problem using the *Picture Options Dialog Box* (p. 2155) (*Figure 31.70* (p. 1465)). Display this dialog box by clicking the **Picture...** button in the *Video Control Dialog Box* (p. 2148).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.70 The Picture Options Dialog Box



Color

Use these controls to ensure that all colors fall into the proper color space for your video device. Also, for best results, set the saturation and brightness levels to 80% or less.

Window Size

If you have a scan converter that converts a portion of the computer screen, you can set the graphics window to a particular pixel size to match the scan converter's window size. You can also create a margin around the picture in the window to keep unwanted parts of the screen (such as the window border) out of the video image.

31.8.3.5. Make Sure Your Tape is Formatted (Preblacked)

Before you can start the recording session, you need to make sure the tape has been preblacked with a time code or frame code. When you start with a brand new tape, you need to take time out and preblack the whole tape first. This can be done using the following procedure:

1. Rewind the tape to the beginning.
2. Select a preblack recording session by clicking on the **Preblack** radio button in the *Video Control Dialog Box* (p. 2148).
3. Send the VTR a black video signal using a scan converter or a picture by clicking on the **Preview** button in the *Animation Recording Options Dialog Box* (p. 2153).
4. Click the **Preblack** button in the **VTR Controls** section of the *Video Control Dialog Box* (p. 2148) to start the preblacking.

31.8.3.6. Start the Recording Session

Make sure you have the proper recording session selected. If you are recording an animation, the **Animation** radio button should be selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To start recording onto tape, you must first go to the "in point" on tape where you want the recording to begin. With a blank tape, it is important to start at about 20 seconds into the tape, so the VTR has a chance to preroll up to the in point. You can use the VTR button controls to position the tape, but an easier way to go to a certain point is to type the time code or frame code in the **Time** or **Frame** counter and press the `<Enter>` key. For example, a time code of `00:02:36:07` is 2 minutes, 36 seconds, and 7 frames. In order to go to this position on the tape, you can enter the time code as `2:36:07`, leaving out the leading zeros, or you can simply enter `23607`, leaving out the leading zeros and colons.

Once your tape is at the start position for your recording session, click the **Record** button to start recording.

31.9. Histogram and XY Plots

In addition to the many graphics tools already discussed, ANSYS FLUENT also provides tools that allow you to generate XY plots and histograms of solution, file, profile, and residual data. You can modify the colors, titles, legend, and axis and curve attributes to customize your plots. The following sections describe the XY and histogram plotting features in ANSYS FLUENT.

31.9.1. Plot Types
31.9.2. XY Plots of Solution Data
31.9.3. XY Plots of File Data
31.9.4. XY Plots of Profiles
31.9.5. XY Plots of Circumferential Averages
31.9.6. XY Plot File Format
31.9.7. Residual Plots
31.9.8. Histograms
31.9.9. Modifying Axis Attributes
31.9.10. Modifying Curve Attributes

31.9.1. Plot Types

Data can be plotted in XY (abscissa/ordinate) form or histogram form. Each form is described below.

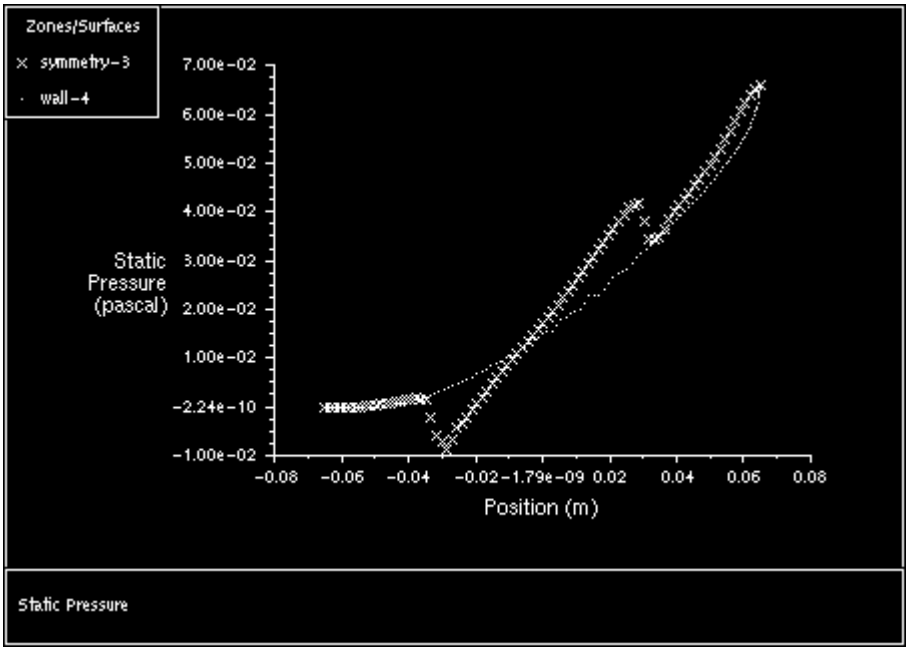
31.9.1.1. XY Plots

An XY (abscissa/ordinate) plot is a line and/or symbol chart of data. Virtually any defined variable or function is accessible for this type of plot. Furthermore, you may read in an externally-generated data file in order to compare your results with experimental data. You can also use the XY-plot facility to plot out profile data, the residual histories of variables, or the time histories if you have a transient problem.

ANSYS FLUENT provides tools for controlling many aspects of the XY plot, including background color, legend, and axis and curve attributes. *Figure 31.71* (p. 1467) shows a sample XY plot.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.71 Sample XY Plot



To differentiate the data being displayed, you can customize the pattern, color and weight of the data lines and the shape, color, and size of the data markers.

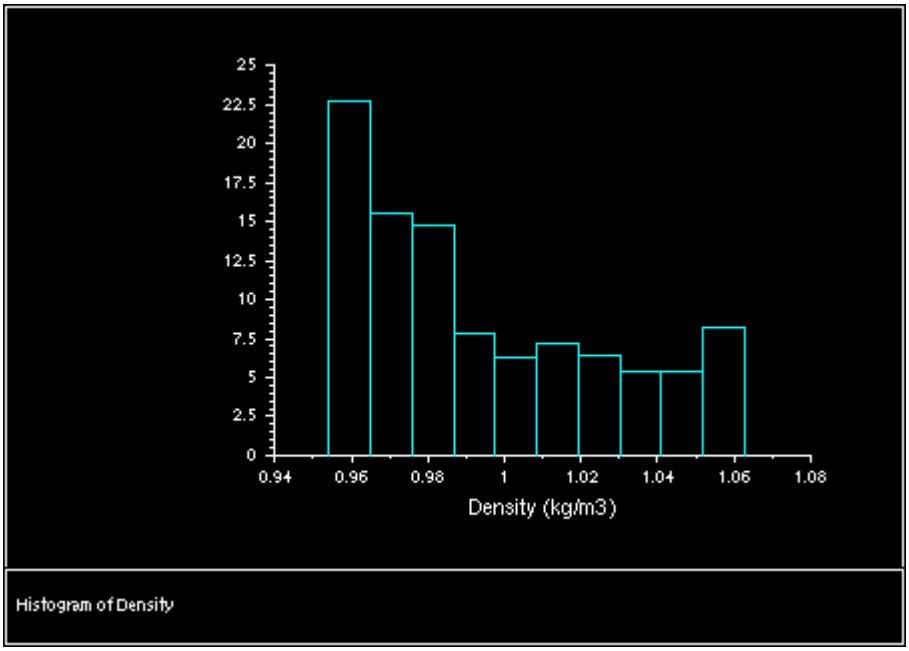
When an XY plot is displayed in the graphics window, you can either use the **mouse-probe** or **mouse-annotate** functions to add text annotations to the plot. All other functions are inactive for XY plots. For more information about the mouse-annotate function, see *Adding Text Using the **Mouse-Annotate** Function* (p. 1416). In addition, you can use any of the mouse buttons to move and resize the legend box.

31.9.1.2. Histograms

A histogram plot is a bar chart of data. It is a representation of a frequency of distribution by means of rectangles of widths representing class intervals and with areas proportional to the corresponding frequencies. When a histogram plot is displayed in the graphics window, you can either use the **mouse-probe** or **mouse-annotate** functions to add text annotations to the plot. All other functions are inactive for histogram plots. (See *Adding Text Using the **Mouse-Annotate** Function* (p. 1416) for more information about the mouse-annotate function.) *Figure 31.72* (p. 1468) shows a sample histogram.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.72 Sample Histogram



See *Histogram Reports* (p. 1522) for information about printing histogram reports. For more information on histogram plots, see *Histograms* (p. 1478).

31.9.2. XY Plots of Solution Data

You can produce a very sophisticated XY plot by using data from several zones, surfaces, or files and modifying the axis and curve attributes. Using the capability for loading external data files, you can create plots that compare your ANSYS FLUENT results with data from other sources. To get further information about the solution, you can investigate the frequency of distribution of the data using a histogram (see *Histograms* (p. 1478)).

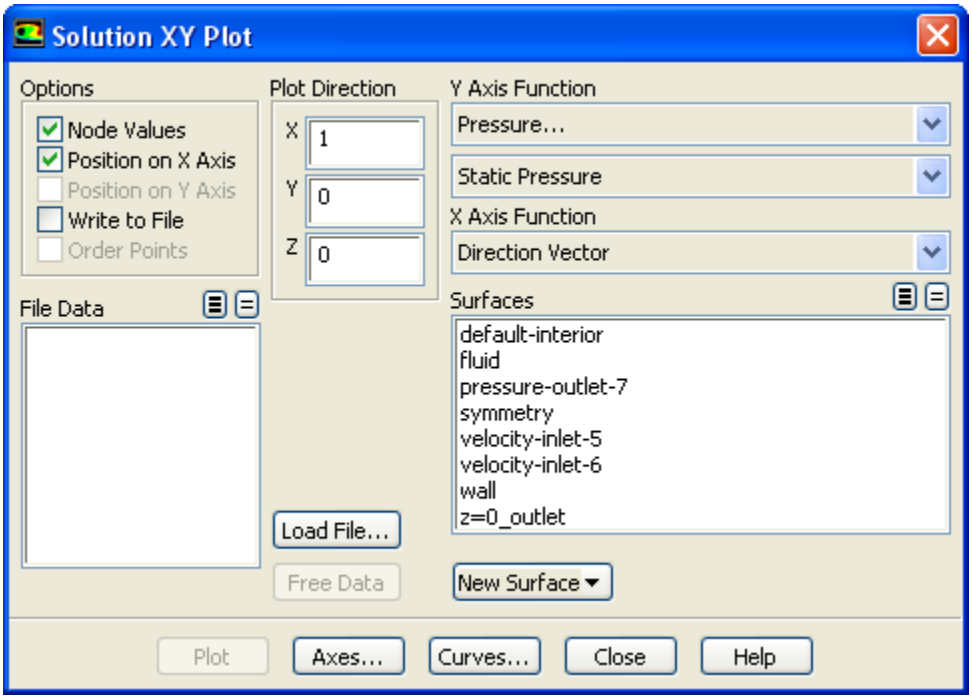
31.9.2.1. Steps for Generating Solution XY Plots

You can create an XY plot of solution data using the *Solution XY Plot Dialog Box* (p. 2012) (*Figure 31.73* (p. 1469)).

Plots → **XY Plot** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.73 The Solution XY Plot Dialog Box



The basic steps for generating a solution XY plot are as follows:

1. Specify the variables(s) you are plotting:
   - To plot a variable on the y axis as a function of position on the x axis, enable the **Position on X Axis** option and choose the variable to be plotted on the y axis in the **Y Axis Function** drop-down list. Select a category from the upper list and then choose the desired quantity in the lower list. For an explanation of the variables in the list, see *Field Function Definitions* (p. 1529).
   - To plot a variable on the x axis as a function of position on the y axis, enable the **Position on Y Axis** option and choose the variable to be plotted on the x axis in the **X Axis Function** drop-down list.
   - To plot one variable as a function of another, turn off both the **Position on X Axis** and **Position on Y Axis** options and select the variables to be plotted in the **X Axis Function** and **Y Axis Function** drop-down lists.
2. Specify the plot direction:
   - To plot a variable as a function of position along a specified direction vector, select **Direction Vector** in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), and specify the components of the direction vector for plotting under **Plot Direction**. The position axis of the plot is indicated by the selection of **Position on X Axis** or **Position on Y Axis**. The positions plotted will have coordinate values that correspond to the dot product of the data coordinate vector with the plot direction vector. For example, if you are plotting a variable at the pressure outlet of the geometry shown in *Figure 31.74* (p. 1470), you would specify the **Plot Direction** vector (1,0,0) since you are interested in how the variable changes as a function of x. *Figure 31.75* (p. 1471) shows the resulting XY plot. (If you specified (0,1,0) as the plot direction, all variable values would be plotted at the same position (see *Figure 31.76* (p. 1471)), since the y value is the same at every point on the pressure outlet.)
   - It is also possible to plot a variable as a function of position along the length of a specified curvilinear surface. The curvilinear surface must be piecewise linear and it cannot contain more than

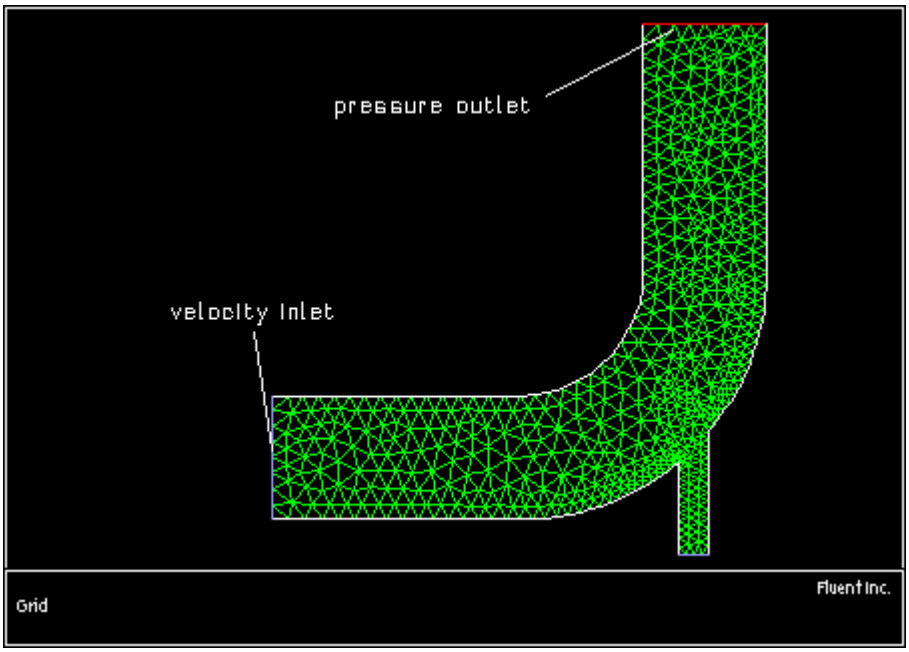
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

one closed curve, such as a complete circle. To plot a variable in this way, select **Curve Length** in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis). Then specify the plot direction along the surface: to plot the variable along the direction of increasing curve length, select **Default** under **Plot Direction**; to plot the variable in the direction of decreasing surface length, select **Reverse**. To check the direction in which the variable will be plotted along a surface, select the surface in the **Surfaces** list and click **Show** under **Plot Direction**. ANSYS FLUENT will display the selected surface in the graphics window, marking the start of the surface with a blue dot and the end of the surface with a red dot. ANSYS FLUENT will also display arrows on the surface showing the direction in which the variable will be plotted.

3. Choose the surface(s) on which to plot data in the **Surfaces** list. Note that if you are plotting a variable as a function of position along the length of a curvilinear surface, you can select only one surface in the **Surfaces** list.
4. Set any of the options described below, or modify the attributes of the axes or curves as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).
5. Click the **Plot** button to generate the XY plot in the active graphics window.

You can use any of the mouse buttons to annotate the XY plot (see *Adding Text to the Graphics Window* (p. 1415)) or move the plot legend from its default position in the upper left corner of the graphics window.

Figure 31.74 Geometry Used for XY Plot



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.75 Data Plotted at Outlet Using a Plot Direction of (1,0,0)

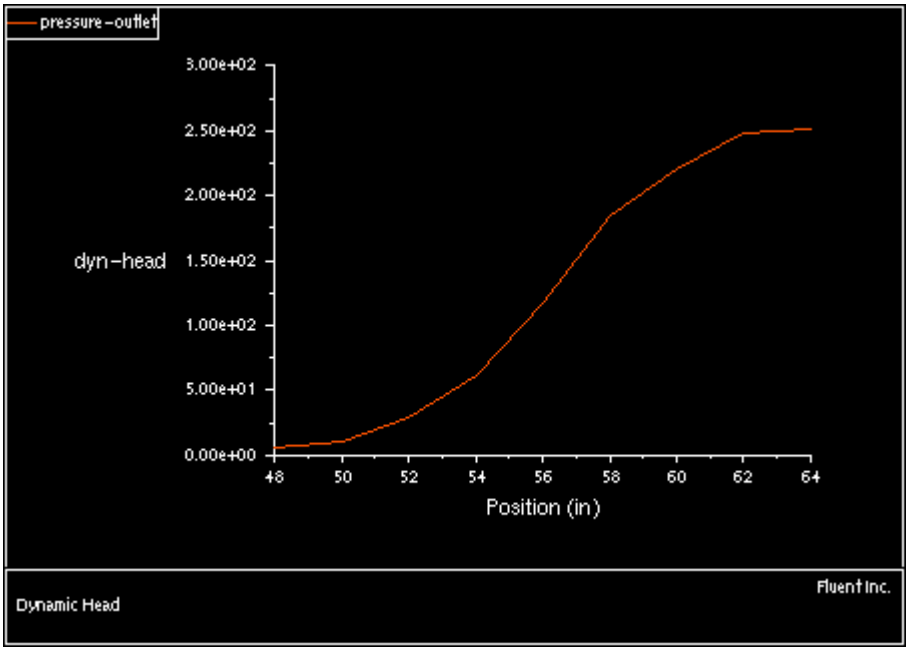
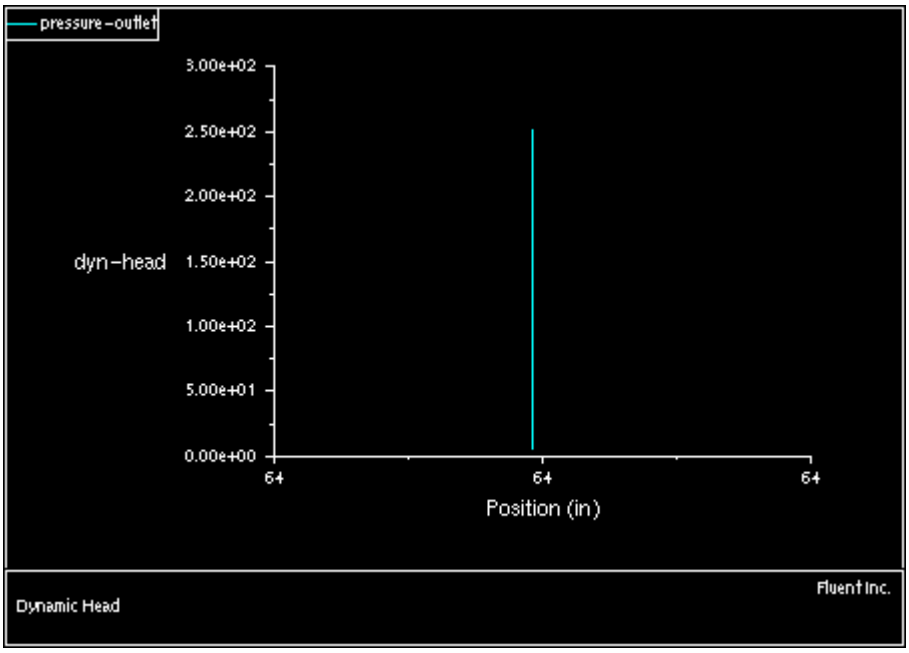


Figure 31.76 Data Plotted at Outlet Using a Plot Direction of (0,1,0)



31.9.2.2. Options for Solution XY Plots

The options mentioned in *Steps for Generating Solution XY Plots* (p. 1468) include the following.

- You can include data from an external file in the solution XY plot to compare your results with experimental data.
- You can also choose node or cell values to be plotted, and save the plot data to a file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.2.2.1. Including External Data in the Solution XY Plot

To add external data to your XY plot for comparison with your results, you must first ensure that any external data files are in the format described in *XY Plot File Format* (p. 1477). You can then load the file(s) by clicking on the **Load File...** button and specifying the file(s) to be read in *The **Select File** Dialog Box* (p. 33). Once a file has been loaded, its title will appear in the **File Data** list. You can choose the data file(s) to be included in your plot from the titles in this list.

To remove a file from the **File Data** list, select it and then click the **Free Data** button.

31.9.2.2.2. Choosing Node or Cell Values

In ANSYS FLUENT you can choose to display the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are displayed. If you prefer to display the cell values, turn the **Node Values** option off. Node-averaged data curves may be somewhat smoother than curves for cell values.

For face-only functions (e.g., **Wall Shear Stress**), the cell values that are displayed for boundary zone surfaces will actually be the face values. These face values are more accurate, as face-only functions are computed on the faces and not on the cells. For these face-only functions, the cell values on postprocessing surfaces will display the values in the cell. For more information about cell values, see *Cell Values* (p. 1529).

If you are displaying the XY plot to show the effect of a porous medium or fan, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

31.9.2.2.3. Saving the Plot Data to a File

Once you have generated an XY plot, you may want to save the plot data to a file. You can read this file into ANSYS FLUENT at a later time and plot it alone using the *File XY Plot Dialog Box* (p. 2016), as described in *XY Plots of File Data* (p. 1472), or add it to a plot of solution data, as described above.

To save the plot data to a file, enable the **Write to File** option in the *Solution XY Plot Dialog Box* (p. 2012). The **Plot** button will change to the **Write...** button. Clicking on the **Write...** button will invoke *The **Select File** Dialog Box* (p. 33), in which you can specify a name and save a file containing the plot data. The format of this file is described in *XY Plot File Format* (p. 1477).

To sort the saved plot data in order of ascending x axis value, enable the **Order Points** option in the *Solution XY Plot Dialog Box* (p. 2012) before you click the **Write...** button. This option is available only when the **Write to File** option is turned on.

31.9.3. XY Plots of File Data

You can produce XY plots using data contained in external files. The *File XY Plot Dialog Box* (p. 2016) allows you to display data read from external files in an abscissa/ordinate plot form. The format of the plot file is described in *XY Plot File Format* (p. 1477).

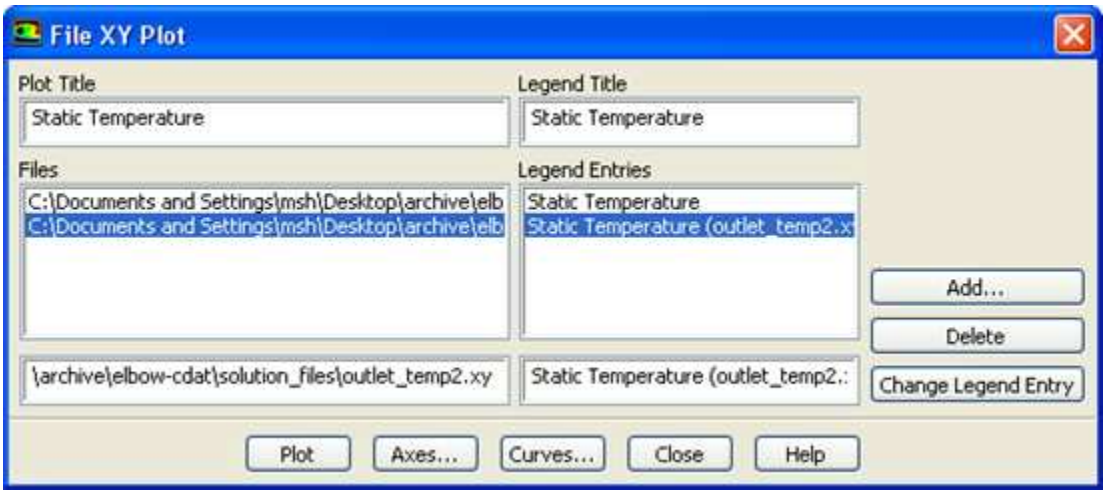
31.9.3.1. Steps for Generating XY Plots of Data in External Files

You can create an XY plot of data contained in one or more external files using the *File XY Plot Dialog Box* (p. 2016) (*Figure 31.77* (p. 1473)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plots → **File** → **Set Up...**

Figure 31.77 The File XY Plot Dialog Box



The steps for generating a file XY plot are as follows:

1. Load each external data file (with the format described in *XY Plot File Format* (p. 1477)) by entering its name in the text field beneath the **Files** list and clicking on the **Add...** button (or pressing <Enter>).If you click **Add...** without specifying a name under **Files** (or if you specify an incorrect or duplicate name), *The **Select File** Dialog Box* (p. 33) will appear and you can specify one or more files there. When a file is loaded, its name will appear in the **Files** list and its title will appear in the **Legend Entries** list. Data in all loaded files will be plotted, so if you decide not to include one of the loaded files in the plot you must select it and click the **Delete** button to remove it.
2. Set any of the options described below, or modify the attributes of the axes or curves as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).
3. Click the **Plot** button to generate an XY plot of the data associated with all loaded files.

31.9.3.2. Options for File XY Plots

The options mentioned in the procedure above include the following. You can change the plot title, legend title, or legend entry.

31.9.3.2.1. Changing the Plot Title

The plot title will appear in the caption box at the bottom of the graphics window. You can modify the plot title by changing the entry in the **Plot Title** text box in the *File XY Plot Dialog Box* (p. 2016) (or by editing the caption box manually, as described in *Changing the Legend Display* (p. 1413)).

31.9.3.2.2. Changing the Legend Entry

When you plot data from a single file, the y axis of the plot will be labeled by the "legend entry." To modify this label, click on the text in the **Legend Entries** list, edit the text that appears in the text field below the list, and then click the **Change Legend Entry** button (or hit <Enter>). When you next plot the data, the new legend entry will appear in the plot.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.3.2.3. Changing the Legend Title

When you plot data from more than one file, a legend will appear in the upper left corner of the graphics window. By default, the legend will have no title. If you want to add a title, enter it in the **Legend Title** text field. The title will appear above the legend the next time you plot the data.

Note that you can use any of the mouse buttons to annotate the plot (see *Adding Text to the Graphics Window* (p. 1415)) or move the legend from its default position.

31.9.4. XY Plots of Profiles

ANSYS FLUENT allows two options for generating XY plots of data related to boundary profiles. Using the *Plot Profile Data Dialog Box* (p. 2017), you can plot the original data points from the profile file you have read into ANSYS FLUENT. Alternatively, you can plot the values assigned to the cell faces on the boundary after the profile file has been interpolated, by using the *Plot Interpolated Data Dialog Box* (p. 2018).

Important

Note that you must have valid data when trying to use the profile plotting options.

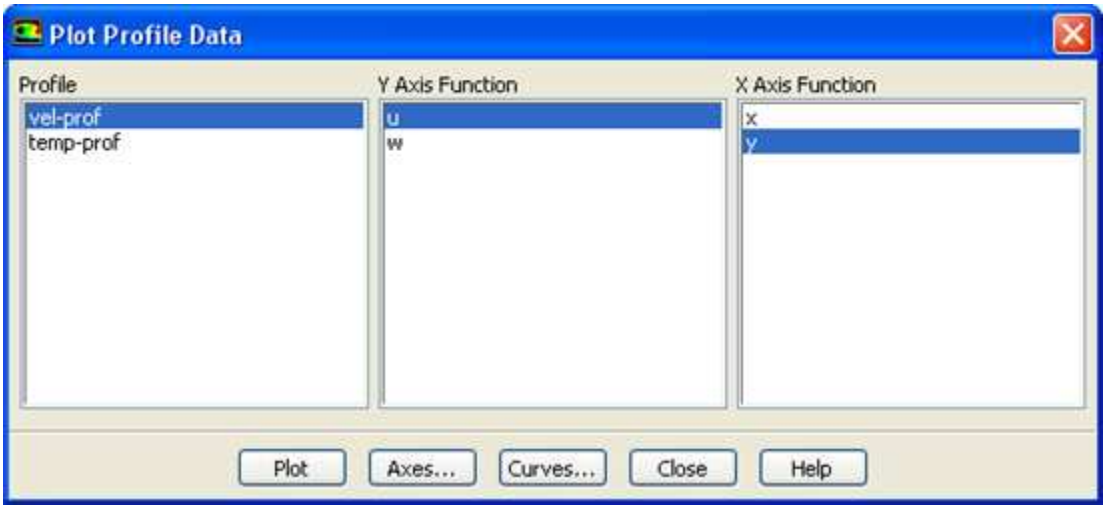
For more information about boundary profiles, see *Profiles* (p. 366).

31.9.4.1. Steps for Generating Plots of Profile Data

Once you have read a profile file, it is available for plotting by using the *Plot Profile Data Dialog Box* (p. 2017).

Plots → **Profile Data** → **Set Up...**

Figure 31.78 The Plot Profile Data Dialog Box



The procedure for generating a XY plot of the original profile data is as follows:

1. Select one of the profiles you have read from the **Profile** selection list.
2. Select a field of the profile from the **Y Axis Function** selection list.
3. Choose a variable against which you want to plot the field data, and select it from the **X Axis Function** selection list. The available variables will vary depending on the profile, and include **x**, **y**, **z**, **r**, and **time**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

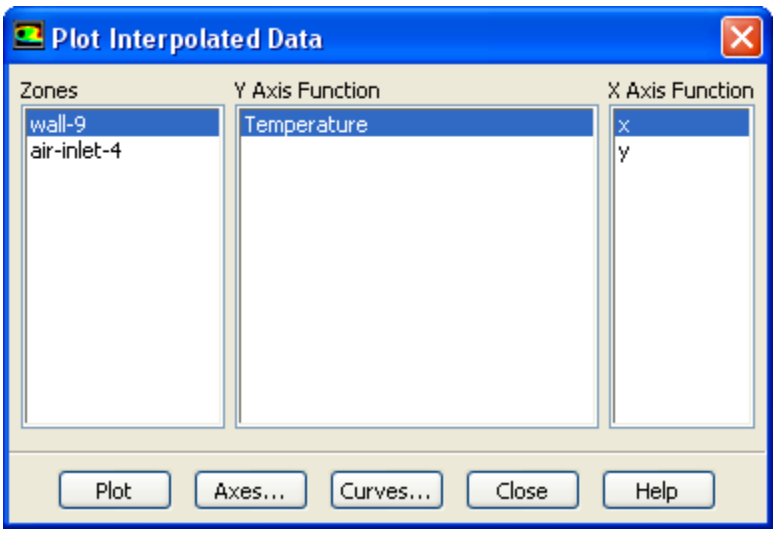
4. Modify the attributes of the axes or curves as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).
5. Click the **Plot** button to generate an XY plot of the profile field data.

31.9.4.2. Steps for Generating Plots of Interpolated Profile Data

To interpolate a profile you must first read a profile file for the case, and select a profile field in a boundary conditions dialog box (e.g., the *Velocity Inlet Dialog Box* (p. 1860)). After the flow solution has been initialized, the cell face values of the boundary zone can be plotted by using the *Plot Interpolated Data Dialog Box* (p. 2018).

Plots → **Interpolated Data** → **Set Up...**

Figure 31.79 The Plot Interpolated Data Dialog Box



The procedure for generating an XY plot of the interpolated data is as follows:

1. Select a zone from the **Zones** selection list. Only the zones for which you have set a profile field as one or more of the parameters will be available in this list.
2. Select a profile-related parameter of the zone from the **Y Axis Function** selection list. The name of the parameter will be the same as that of the drop-down list in the boundary condition dialog box from which the profile field was selected.
3. Choose a variable against which you want to plot the field data, and select it from the **X Axis Function** selection list. The available variables are **x**, **y**, and (for 3D cases) **z**.
4. Modify the attributes of the axes or curves as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).
5. Click the **Plot** button to generate an XY plot of the cell face values on the boundary.

31.9.5. XY Plots of Circumferential Averages

You can also generate a plot of circumferential averages in ANSYS FLUENT. This allows you to find the average value of a quantity at several different radial or axial positions in your model. ANSYS FLUENT computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.5.1. Steps for Generating an XY Plot of Circumferential Averages

You can generate an XY plot of circumferential averages in the radial direction using the `circum-avg-radial` text command:

`plot` → `circum-avg-radial`

or you can use the `circum-avg-axial` text command to generate an average in the axial direction:

`plot` → `circum-avg-axial`

The procedure for generating an XY plot of circumferential averages is as follows:

1. Specify the variable to be averaged by typing its name when ANSYS FLUENT prompts you for `averages of`. You can press `<Enter>` to see a list of available variables.
2. Choose the surface on which to plot data by typing its name when ANSYS FLUENT prompts you for `on surface`.

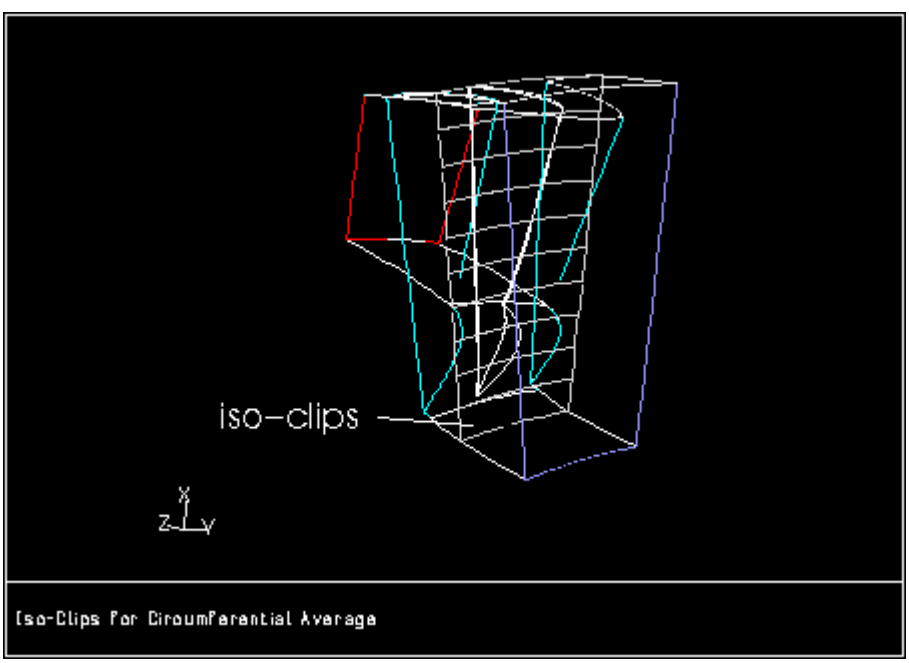
   Important

   Use the *Mesh Display Dialog Box* (p. 1635) to see a list of surfaces on which you can plot data. Pressing `<Enter>` will not show a list of available surfaces.

3. Specify the `number of bands` to be created. The default number of bands is 5.

   ANSYS FLUENT will create circumferential bands by isoclipping the specified surface into equal bands of radial or axial coordinate. An example of the iso-clips created is shown in *Figure 31.80* (p. 1476). The radial or axial coordinate is derived from the rotation axis of the **Reference Zone** specified in the *Reference Values Task Page* (p. 1898).

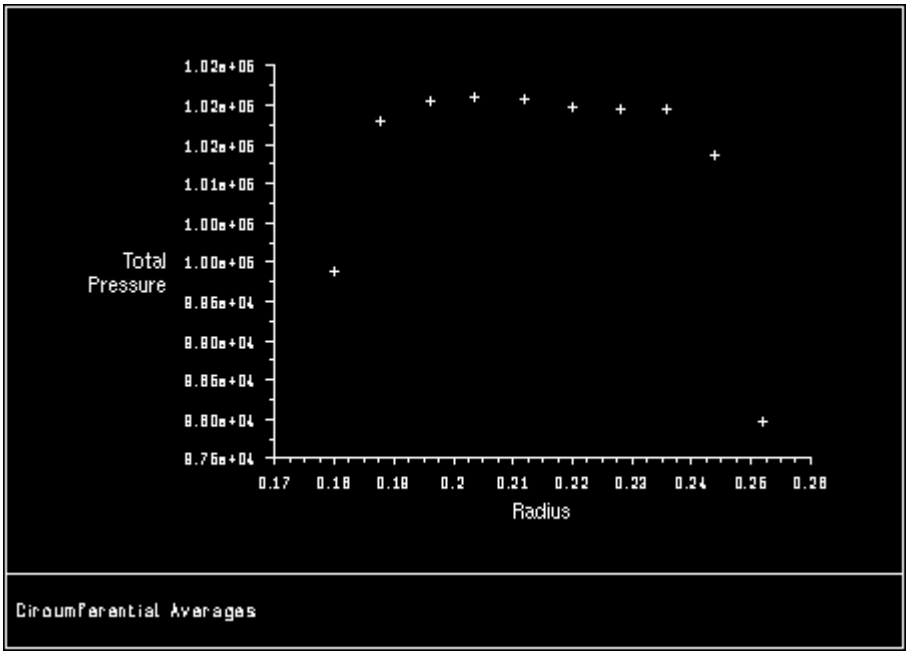
Figure 31.80 Iso-Clips Created For Circumferential Averaging



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT then computes the average of the variable for each band using the area-weighted average described in Computing Surface Integrals of the Theory Guide. Finally, it plots the average of the variable as a function of radial or axial coordinate. *Figure 31.81* (p. 1477) shows an example of an XY plot of circumferential averages using radial coordinates.

Figure 31.81 XY Plot of Circumferential Averages



When the circumferential average plot is generated, ANSYS FLUENT also creates a new surface called `radial-bands` or `axial-bands`, which contains the iso-clips described above (see *Figure 31.80* (p. 1476)). You can use this surface to generate other XY plots. For more information on the creation and manipulation of surfaces, see *Creating Surfaces for Displaying and Reporting Data* (p. 1355).

31.9.5.2. Customizing the Appearance of the Plot

If you want to customize the appearance of the axes or curves in a circumferential average plot, you can save the plot data to a file (using the `plot-to-file` text command, as described in *XY Plot File Format* (p. 1477)), read the file into ANSYS FLUENT and plot it again (using the *File XY Plot Dialog Box* (p. 2016), as described in *XY Plots of File Data* (p. 1472)), and then use the *Axes Dialog Box* (p. 2022) and *Curves Dialog Box* (p. 2024) (as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482)) to modify the appearance of the plot.

To save the plot data to a file, first use the `plot-to-file` text command to specify the name of the file.

`plot` → `file-set` → `plot-to-file`

Then generate the circumferential average XY plot as described above. ANSYS FLUENT will display the plot in the graphics window, and also save the plot data to the specified file.

31.9.6. XY Plot File Format

The XY file format read or written by ANSYS FLUENT includes the following information:

- The title of the plot

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The label for the abscissa and the ordinate
- Cortex variables and pairs of abscissa/ordinate data for each curve in the plot

The following sample file illustrates the XY file format:

```
(title "Velocity Magnitude")
(labels "Position" "Velocity Magnitude")

((xy/key/label "pressure-inlet-8")
(xy/key/visible? #t)
(xy/line/pattern "--")
0.0000 230.097 0.
0625 160.551
0.1250 149.205
...
0.5000 183.007
)
```

Similar to the case file format, parentheses bound the various pieces of information in the formatted, ASCII file. The title (`title " "`) and labels (`labels " "`) must be first in the file, then each curve has information in the form (`(cxvar value) x y x y x y...`), where there may be zero or more Cortex variables defined for each curve.

You do not have to include Cortex variables to import your XY data. For example, you may wish to import experimental data to compare with the ANSYS FLUENT solution. The following example would use the default Cortex variables in the code to define the data. After you import the file into ANSYS FLUENT, you could then use the *Axes Dialog Box* (p. 2022) and the *Curves Dialog Box* (p. 2024) to customize the XY plot, as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).

```
(title "Experiment, Run 11")
(labels "X, m" "Cp")
( 0 1.5
 1.5 1.3
 3.2 1.5
 5.1 1.2
)
```

31.9.7. Residual Plots

Residual history can be displayed using an XY plot. The abscissa of the plot corresponds to the number of iterations and the ordinate corresponds to the log-scaled residual values.

To plot the current residual history, click the **Plot** button in the *Residual Monitors Dialog Box* (p. 1915).

Monitors → **Residuals** → **Edit...**

For additional information about using the *Residual Monitors Dialog Box* (p. 1915) to plot residuals, see *Printing and Plotting Residuals* (p. 1272).

31.9.8. Histograms

Histograms can be displayed in a graphics window using a bar chart (or printed in the console window, as described in *Histogram Reports* (p. 1522)). The abscissa of the chart is the desired solution quantity and the ordinate is the percentage of the total number of cells.

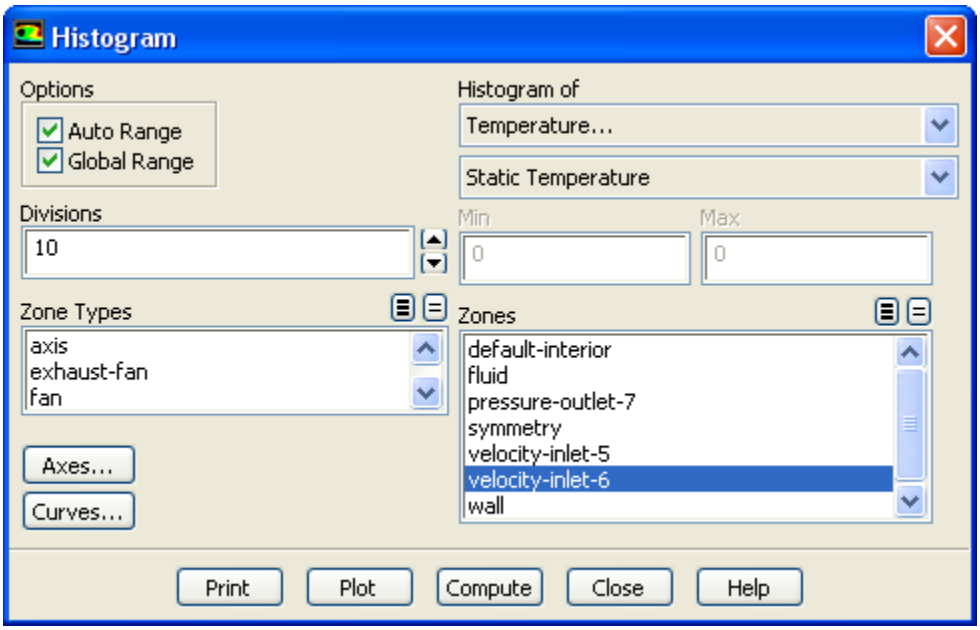
31.9.8.1. Steps for Generating Histogram Plots

You can create a histogram plot of solution data using the *Histogram Dialog Box* (p. 2015) (*Figure 31.82* (p. 1479)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plots → **Histogram** → **Set Up...**

Figure 31.82 The Histogram Dialog Box



The steps for generating a histogram plot are as follows:

1. Choose the scalar quantity to be plotted in the **Histogram Of** drop-down list. Select a category in the upper list and then select the desired quantity in the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
2. Set the number of data intervals that will be plotted in the histogram in the **Divisions** field. By default there will be 10 intervals ("bars") in the histogram plot to finer intervals, increase the number of **Divisions**. You may want to click the **Compute** button to update the **Min** and **Max** fields when you are trying to decide how many divisions to plot.
3. Select the face or cell zone under **Zones** for which you want results plotted or printed. If all zones are selected, then the entire domain will be plotted. You can also plot histograms based on the selected **Zone Types**.
4. Set the option described below, if desired, or modify the attributes of the axes or curves as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).
5. Click the **Plot** button to generate the histogram plot in the active graphics window.
6. Click the **Print** button to print out your histogram results on individual zones, or the entire domain. Similarly, you can click the **Compute** button to calculate your histogram results on individual zones, or the entire domain.

31.9.8.2. Options for Histogram Plots

Other than the axis and curve attribute controls mentioned in the procedure above, the only option for histogram plotting is the ability to specify a subrange of values to be plotted.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.8.2.1. Specifying the Range of Values Plotted

By default, the range of values included in the histogram plot is automatically set to the range of values in the entire domain for the selected variable. If you want to focus in on a smaller range of values, you can restrict the range to be displayed.

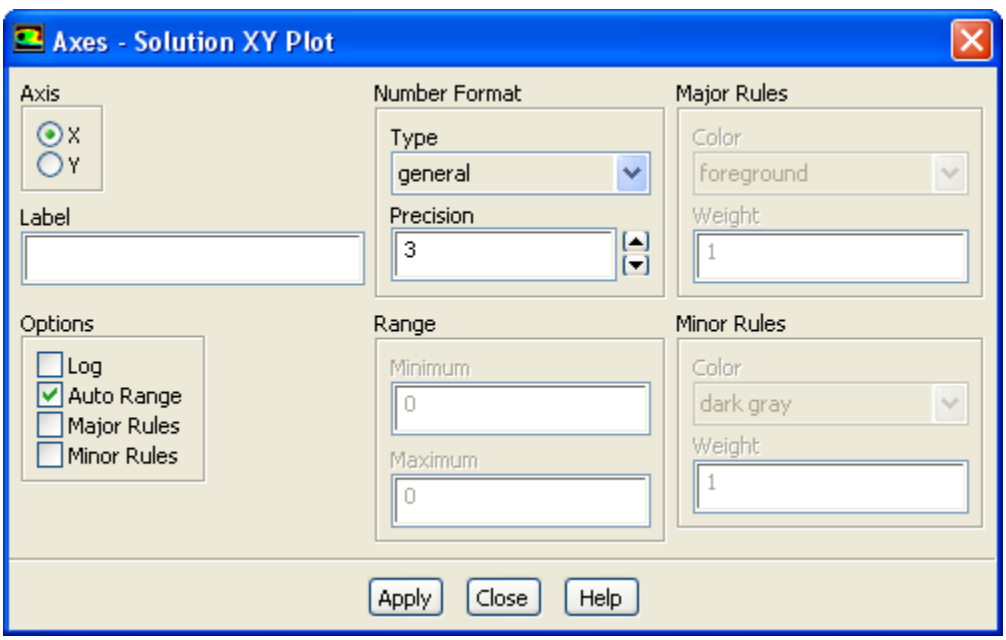
To manually set the range of values, turn off the **Auto Range** option in the *Histogram Dialog Box* (p. 2015). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be plotted. To show the default range at any time, click the **Compute** button and the **Min** and **Max** fields will be updated.

You can also choose to base the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain. To do this, turn off the **Global Range** option in the *Histogram Dialog Box* (p. 2015). The **Min** and **Max** values will be updated when you next click **Compute**.

31.9.9. Modifying Axis Attributes

You can modify the appearance of the XY and parameters that control the labels, scale, range, numbers, and major and minor rules. For each type of plot (solution XY, file XY, profile, residual, histogram, etc.), you can set different axis parameters in the *Axes Dialog Box* (p. 2022) (*Figure 31.83* (p. 1480)). Note that the title following **Axes** in the dialog box indicates which plot environment you are changing (e.g., the **Axes - Solution XY Plot** dialog box controls parameters for solution XY plots).

Figure 31.83 The Axes Dialog Box



To open the *Axes Dialog Box* (p. 2022) for a particular plot type, click the **Axes...** button in the appropriate dialog box (e.g., the **Solution XY Plot**, **File XY Plot**, **Plot Profile Data**, **Plot Interpolated Data**, or **Residual Monitors** dialog box).

31.9.9.1. Using the Axes Dialog Box

The *Axes Dialog Box* (p. 2022) allows you to independently control the characteristics of the ordinate (y axis) and abscissa (x axis) on an XY plot or parameters for one axis or the other, by following the procedure below:

1. Choose the axis for which you want to modify the attributes by selecting **X** or **Y** under **Axis**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the desired parameters.
3. Click **Apply** and then choose the other axis and repeat the steps, if desired.

The changes to the axis attributes will appear in the graphics window the next time you generate a plot.

31.9.9.1.1. Changing the Axis Label

If you want to modify the label for the axis, you can do so by editing the **Label** text field in the *Axes Dialog Box* (p. 2022).

31.9.9.1.2. Changing the Format of the Data Labels

You can change the format of the labels that define the primary data divisions on the axes using the controls under the **Number Format** heading in the *Axes Dialog Box* (p. 2022).

- To display the real value with an integral and fractional part (e.g., 1.0000), select **float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.
- To display the real value with a mantissa and exponent (e.g., 1.0e-02), select **exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.
- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **general** in the **Type** drop-down list.

31.9.9.1.3. Choosing Logarithmic or Decimal Scaling

By default, decimal scaling is used for both axes (except for the y axis in residual plots, which uses a log scale). If you want to change to a logarithmic scale, enable the **Log** option in the *Axes Dialog Box* (p. 2022). To return to a decimal scale, turn off the **Log** option. Note that when you are using the logarithmic scale, the **Range** values are the exponents; to specify a logarithmic range from 1 to 10000, for example, you will specify a minimum value of 1 and a maximum value of 4.

31.9.9.1.4. Resetting the Range of the Axis

By default, the extents of the axis will range from the minimum value plotted to the maximum value plotted. If you want to change the range or extents of the axis, you can do so by turning off the **Auto Range** option in the *Axes Dialog Box* (p. 2022) and setting the new **Minimum** and **Maximum** values for the **Range**. This feature is useful when you are generating a series of plots and you want the extents of one or both of the axes to be the same, even if the range of plotted values differs. For example, if you are generating plots of temperature on several different wall zones, you might want the minimum and maximum temperature on the y axis to be the same in every plot so that you can easily compare one plot with another. You would determine a temperature range that includes the temperatures on all walls, and use that as the range for the y axis in each plot.

31.9.9.1.5. Controlling the Major and Minor Rules

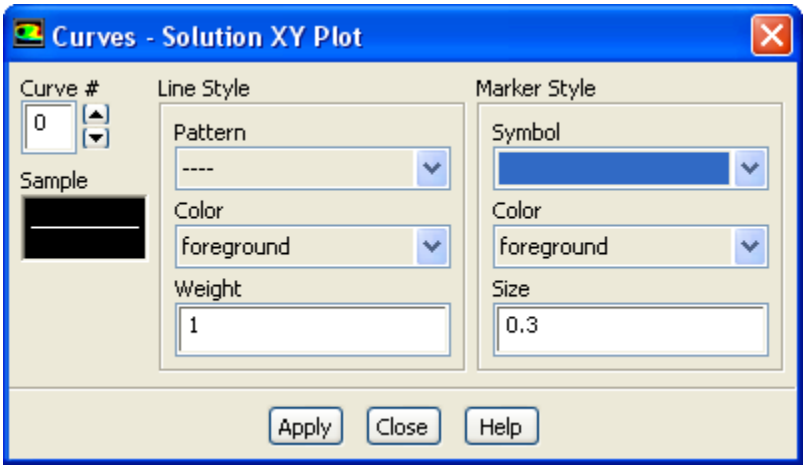
ANSYS FLUENT allows you to display major and/or minor rules on the axes. Major and minor rules are the horizontal or vertical lines that mark, respectively, the primary and secondary data divisions and span the whole plot window to produce a "mesh." To add major or minor rules to the plot, enable the **Major Rules** or **Minor Rules** option. You can then specify a color and weight for each type of rule. Under the **Major Rules** or **Minor Rules** heading, select the desired color for the lines in the **Color** drop-down list and specify the line thickness in the **Weight** field. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.10. Modifying Curve Attributes

The data curves in XY plots and histograms can be represented by any combination of lines and markers. You can modify the attributes of the curves, including the patterns, weights, and colors of the lines, and the symbols, sizes, and colors of the markers. For each type of plot (solution XY, file XY, profile, residual, parameters in the *Curves Dialog Box* (p. 2024) (*Figure 31.84* (p. 1482)). Note that the title following **Curves** in the dialog box indicates which plot environment you are changing (e.g., the **Curves - Solution XY Plot** dialog box controls curve parameters for solution XY plots).

Figure 31.84 The Curves Dialog Box



To open the *Curves Dialog Box* (p. 2024) for a particular plot type, click the **Curves...** button in the appropriate dialog box (e.g., **Solution XY Plot**, **File XY Plot**, **Plot Profile Data**, **Plot Interpolated Data**, or **Residual Monitors** dialog box).

31.9.10.1. Using the Curves Dialog Box

The *Curves Dialog Box* (p. 2024) allows you to independently control the characteristics of each data curve in an XY plot or parameters for a curve, you will follow the procedure below:

1. Specify the curve for which you want to modify the attributes by increasing or decreasing the **Curve #** counter. The curves are numbered sequentially, starting from 0. For example, if you were plotting flow-field values on two surfaces, the first surface would be curve 0, and the second, curve 1. If the plot contains only one curve, the **Curve #** is set to 0 and is not editable.
2. Set the desired line and/or marker parameters as described below.
3. Click **Apply**and then choose another **Curve #** and repeat the steps, if desired.

Your changes to the curve attributes will appear in the graphics window the next time you generate a plot.

31.9.10.1.1. Changing the Line Style

You can control the pattern, color, and weight of the line using the controls under the **Line Style** heading:

- To set the line pattern for the curve, choose one of the items in the **Pattern** drop-down list. Except for **center** and **phantom** lines, the list displays examples of the pattern choices. A **center** line alternates a very long dash and a short dash and a **phantom** line alternates a very long dash and a double short dash. Note that selecting the second item in the drop-down list, represented by 4 short dashes, will result in a solid-line curve.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you do not want the data points to be connected by any type of line (i.e., if you plan to use just markers), select the "blank" choice, which is the first item in the **Pattern** list.

- To set the color of the line, pick one of the choices in the **Color** drop-down list.
- To define the line thickness, set the value of **Weight**. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

31.9.10.1.2. Changing the Marker Style

You can control the symbol, color, and size for the data marker using the controls under the **Marker Style** heading:

- To set the symbol used to mark data, choose one of the items in the **Symbol** drop-down list. The list displays examples of the symbol choices. For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol `/*\` (filled-in upward-pointing triangle) could be used for the marker representing the upper surface data, and the symbol `\*/` (filled-in downward-pointing triangle) could be used for the marker representing the lower surface data.

Important

If you do not want the data points to be represented by markers (i.e., if you plan to use just a line connecting the data points), select the "blank" choice, which is the first item in the **Style** list.

- To set the color of the marker, pick one of the choices in the **Color** drop-down list.
- To define the size of the data marker, set the value of **Size**. A symbol of size 1.0 is 3.0% of the height of the display screen, except for the "." symbol, which is always one pixel.

31.9.10.1.3. Previewing the Curve Style

To see what a particular setting will look like in the plot, you can preview it in the **Sample** window of the *Curves Dialog Box* (p. 2024). A single marker and/or line will be shown with the specified style attributes.

31.10. Turbomachinery Postprocessing

In addition to the many graphics tools already discussed, ANSYS FLUENT also provides turbomachinery-specific postprocessing features which can be accessed once you have defined the topology of the problem. Information on postprocessing for turbomachinery applications is provided in the following sections:

31.10.1. Defining the Turbomachinery Topology
31.10.2. Generating Reports of Turbomachinery Data
31.10.3. Displaying Turbomachinery Averaged Contours
31.10.4. Displaying Turbomachinery 2D Contours
31.10.5. Generating Averaged XY Plots of Turbomachinery Solution Data
31.10.6. Globally Setting the Turbomachinery Topology
31.10.7. Turbomachinery-Specific Variables

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.1. Defining the Turbomachinery Topology

In order to establish the turbomachinery-specific coordinate system used in subsequent postprocessing functions, ANSYS FLUENT requires you to define the topology of the flow domain. The procedure for defining the topology is described further in this section, along with details about the boundary types.

The current implementation of the turbomachinery topology definition for postprocessing is no longer limited to one row of blades at a time. If your geometry contains multiple rows of blades, you can define all turbomachinery topologies simultaneously. You can name and/or manage all topologies and perform various turbomachinery postprocessing tasks on a single topology or on all topologies at once.

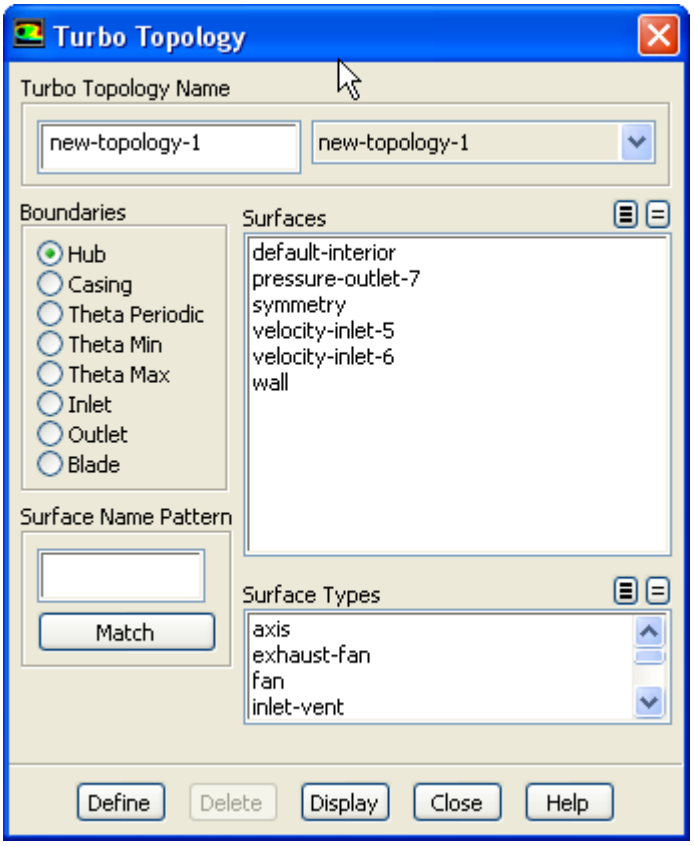
Important

The turbo coordinates can only be generated properly if the correct rotation axis is specified in the boundary conditions dialog box for the fluid zone (see *Specifying the Rotation Axis* (p. 217)).

To define the turbomachinery topology in ANSYS FLUENT, you will use the *Turbo Topology Dialog Box* (p. 2094) (*Figure 31.85* (p. 1484)).

Define → **Turbo Topology...**

Figure 31.85 The Turbo Topology Dialog Box



The procedure for defining topology for your turbomachinery application are as follows:

1. Select a boundary type under **Boundaries** (e.g., **Hub** in *Figure 31.85* (p. 1484)). The boundary types are described in detail in *Boundary Types* (p. 1486).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Surfaces** list, choose the surface(s) that represent the boundary type you selected in step 1.

   If you want to select several surfaces of the same type, you can select that type in the **Surface Types** list instead. All of the surfaces of that type will be selected automatically in the **Surfaces** list (or deselected, if they are all selected already). Another shortcut is to specify a **Surface Name Pattern** and click **Match** to select surfaces with names that match the specified pattern. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (e.g., **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).

3. Repeat the steps 1 and 2 for all the boundary types that are relevant for your model.

   Important

   For a complete turbo topology definition the surfaces defined as inlet, outlet, hub, casing, periodic, theta min, and theta max (if available) should form a closed domain.

4. Enter a name in the **Turbo Topology Name** field or keep the default name.

5. Click **Define** to complete the definition of the boundaries.

   ANSYS FLUENT will inform you that the turbomachinery postprocessing functions have been activated, and the **Turbo** menu will appear in ANSYS FLUENT's menu bar at the top of the console window.

6. Specify a position vector that is defined as $\theta=0$. This position vector should be outside the domain, e.g., if your domain lies in the first and second quadrant, specify negative y axis as the zero θ line. This will ensure that there is no discontinuity in angular coordinates within the domain. This can be done using the `display/set/zero-angle-dir` command.

   Default zero θ line is $+y$ axis. If this axis passes through the domain, you should define the zero θ line, so as to satisfy above criteria.

7. To view a defined topology, select the topology from the **Turbo Topology Name** drop-down list and click **Display**. The defined topology is shown in the active graphics window. This allows you to visually check the boundaries to ensure that you have defined them correctly.

8. To edit a defined topology, select the topology from the **Turbo Topology Name** drop-down list, make the appropriate changes and click **Modify**.

9. To remove a defined topology, select the topology from the **Turbo Topology Name** drop-down list and click **Delete**.

Important

Note that the topology setup that you define will be saved to the case file when you save the current model. Thus, if you read this case back into ANSYS FLUENT, you do not need to set up the topology again.

However, use of a boundary condition file to set the turbo topology for two similar cases may not work properly. In that case you need to set the turbo topology manually.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.1.1. Boundary Types

The boundaries for the turbomachinery topology are as follows (see *Figure 31.86* (p. 1486)):

Hub
is the wall zone(s) forming the lower boundary of the flow passage (generally toward the axis of rotation of the machine).

Casing
is the wall zone(s) forming the upper boundary of the flow passage (away from the axis of rotation of the machine).

Theta Periodic
is the periodic boundary zone(s) on the circumferential boundaries of the flow passage.

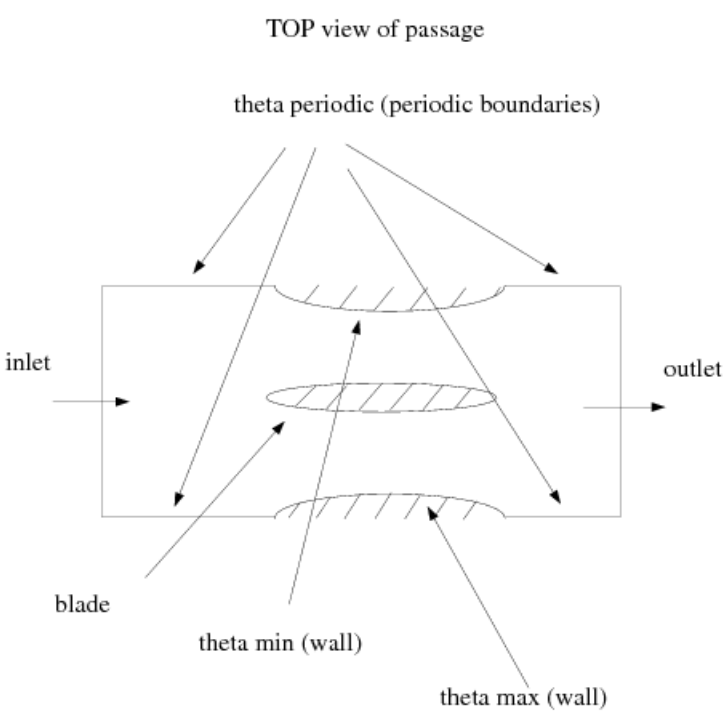
Theta Min
and **Theta Max** are the wall zones at the minimum and maximum angular (θ) positions on a circumferential boundary.

Inlet
is the inlet zone(s) through which the flow enters the passage.

Outlet
is the outlet zone(s) through which the flow exits the passage.

Blade
is the wall zone(s) that defines the blade(s) (if any). Note that these zones cannot be attached to the circumferential boundaries. For this situation, use **Theta Min** and **Theta Max** to define the blade.

Figure 31.86 Turbomachinery Boundary Types



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

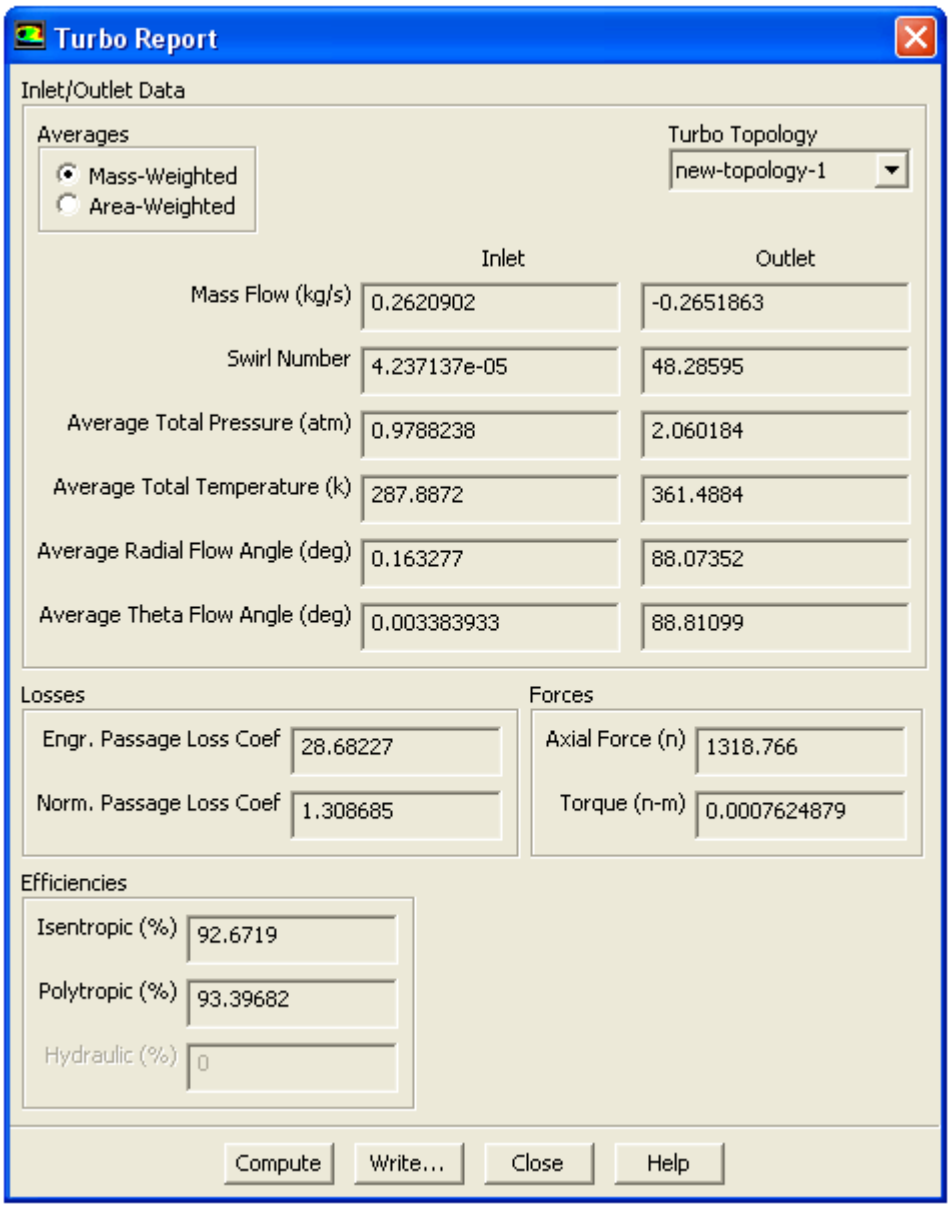
31.10.2. Generating Reports of Turbomachinery Data

Once you have defined your turbomachinery topologies, as described in *Defining the Turbomachinery Topology* (p. 1484), you can report a number of turbomachinery quantities, including mass flow, swirl number, torque, and efficiencies.

To report turbomachinery quantities in ANSYS FLUENT, you will use the *Turbo Report Dialog Box* (p. 2173) (*Figure 31.87* (p. 1487)).

Turbo → **Report...**

Figure 31.87 The Turbo Report Dialog Box



The procedure for using this dialog box is as follows:

1. Under **Averages**, specify whether you want to report **Mass-Weighted** or **Area-Weighted** averages.
2. Under **Turbo Topology**, specify a predefined turbomachinery topology from the drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Compute**.ANSYS FLUENT will compute the turbomachinery quantities as described below, and display their values.
4. If you want to save the reported values to a file, click **Write...** and specify a name for the file in *The **Select File** Dialog Box* (p. 33).

31.10.2.1. Computing Turbomachinery Quantities

31.10.2.1.1. Mass Flow

The mass flow rate through a surface is defined as follows:

$$\dot{m} = \int_A (\rho \vec{v} \cdot \hat{n})\, dA \qquad \textbf{(31–2)}$$

where

A = is the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

31.10.2.1.2. Swirl Number

The swirl number is defined as follows:

$$SW = \frac{\int_S r v_\theta (\vec{v} \cdot \hat{n})\, dS}{\bar{r} \int_S v_z (\vec{v} \cdot \hat{n})\, dS} \qquad \textbf{(31–3)}$$

where

r = the radial coordinate (specifically, the radial distance from the axis of rotation)

v_θ = the tangential velocity

$\vec{v}$ = the velocity vector

$\hat{n}$ = a unit vector normal to the surface,

S = the inlet or outlet

$$\bar{r} = \frac{1}{S} \int_S r\, dS \qquad \textbf{(31–4)}$$

31.10.2.1.3. Average Total Pressure

The area-averaged total pressure is defined as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{p}_t = \frac{\int_A p_t dA}{A} \tag{31–5}$$

where p_t is the total pressure and A is the area of the inlet or outlet.

The mass-averaged total pressure is defined as follows:

$$\overline{p}_t = \frac{\int_A \left(\rho\, p_t |\vec{v} \cdot \hat{n}|\right) dA}{\int_A (\rho\, |\vec{v} \cdot \hat{n}|)\, dA} \tag{31–6}$$

where,

p_t = the total pressure

A = the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

31.10.2.1.4. Average Total Temperature

The area-averaged total temperature is defined as follows:

$$\overline{T}_t = \frac{\int_A T_t dA}{A} \tag{31–7}$$

where T_t is the total temperature and A is the area of the inlet or outlet.

The mass-averaged total temperature is defined as follows:

$$\overline{T}_t = \frac{\int_A \left(\rho\, T_t |\vec{v} \cdot \hat{n}|\right) dA}{\int_A (\rho\, |\vec{v} \cdot \hat{n}|)\, dA} \tag{31–8}$$

where,

T_t = the total temperature

A = the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

31.10.2.1.5. Average Flow Angles

The area-averaged flow angles are defined as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\bar{\alpha}_r = \tan^{-1}\left(\frac{\int_A v_\theta dA}{\int_A v_z dA}\right) \tag{31–9}$$

in the radial direction, and

$$\bar{\alpha}_\theta = \tan^{-1}\left(\frac{\int_A v_r dA}{\int_A v_z dA}\right) \tag{31–10}$$

in the tangential direction, where v_z, v_r, and v_θ represent the axial, radial, and tangential velocities, respectively.

The mass-averaged flow angles are defined as follows:

$$\bar{\alpha}_{r,m} = \tan^{-1}\left(\frac{\int_A (\rho v_r) dA}{\int_A (\rho v_z) dA}\right) \tag{31–11}$$

in the radial direction, and

$$\bar{\alpha}_{\theta,m} = \tan^{-1}\left(\frac{\int_A (\rho v_\theta) dA}{\int_A (\rho v_z) dA}\right) \tag{31–12}$$

in the tangential direction.

31.10.2.1.6. Passage Loss Coefficient

The engineering loss coefficient is defined as follows:

$$K_L = \frac{\bar{p}_{t,i} - \bar{p}_{t,o}}{\frac{1}{2}\rho \bar{v}_i^2} \tag{31–13}$$

where,

$\bar{p}_{t,i}$ = the mass-averaged total pressure at the inlet

$\bar{p}_{t,o}$ = the mass-averaged total pressure at the outlet

ρ = the density of the fluid

$\bar{v}_i$ = the mass-averaged velocity magnitude at the inlet

The normalized loss coefficient is defined as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$K_{L,n} = \frac{\overline{p}_{t,i} - \overline{p}_{t,o}}{\overline{p}_{t,i} - \overline{p}_{s,o}} \quad \textbf{(31–14)}$$

where $\overline{p}_{s,o}$ is the mass-averaged static pressure at the outlet.

31.10.2.1.7. Axial Force

The axial force on the rotating parts is defined as follows:

$$F_a = \left(\int_S (\overline{\overline{\tau}} \cdot \hat{n}) dS \right) \cdot \hat{a} \quad \textbf{(31–15)}$$

where,

S = the surfaces comprising all rotating parts

$\overline{\overline{\tau}}$ = the total stress tensor (pressure and viscous stresses)

$\hat{n}$ = a unit vector normal to the surface

$\hat{a}$ = a unit vector parallel to the axis of rotation

31.10.2.1.8. Torque

The torque on the rotating parts is defined as follows:

$$T = \left(\int_S (\vec{r} \times (\overline{\overline{\tau}} \cdot \hat{n})) dS \right) \cdot \hat{a} \quad \textbf{(31–16)}$$

where,

S = the surfaces comprising all rotating parts

$\overline{\overline{\tau}}$ = the total stress tensor

$\hat{n}$ = a unit vector normal to the surface

$\vec{r}$ = the position vector

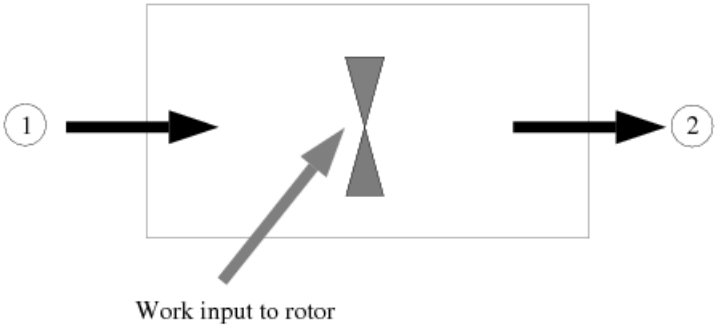
$\hat{a}$ = a unit vector parallel to the axis of rotation

31.10.2.1.9. Efficiencies for Pumps and Compressors

The definitions of the efficiencies for compressible and incompressible flows in pumps and compressors are described in this section. Efficiencies for turbines are described later in this section. Consider a pumping or compression device operating between states 1 and 2 as illustrated in *Figure 31.88* (p. 1492). Work input to the device is required to achieve a specified compression of the working fluid.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.88 Pump or Compressor



Assuming that the processes are steady state, steady flow, and that the mass flow rates are equal at the inlet and outlet of the device (no film cooling, bleed air removal, etc.), the efficiencies for incompressible and compressible flows are as described in the following subsections.

31.10.2.1.9.1. Incompressible Flows

For devices such as liquid pumps and fans at low speeds, the working fluid can be treated as incompressible. The efficiency of a pumping process with an incompressible working fluid is defined as the ratio of the head rise achieved by the fluid to the power supplied to the rotor/impeller. This can be expressed as follows:

$$\eta = \frac{Q\left(p_{t2} - p_{t1}\right)}{T\omega} \qquad \textbf{(31–17)}$$

where,

Q = volumetric flow rate

p_t = total pressure

T = net torque acting on the rotor/impeller

ω = rotational speed

This definition is sometimes called the "hydraulic efficiency". Often, other efficiencies are included to account for flow leakage (volumetric efficiency) and mechanical losses along the transmission system between the rotor and the machine providing the power for the rotor/impeller (mechanical efficiency). Incorporating these losses then yields a total efficiency for the system.

31.10.2.1.9.2. Compressible Flows

For gas compressors that operate at high speeds and high pressure ratios, the compressibility of the working fluid must be taken into account. The efficiency of a compression process with a compressible working fluid is defined as the ratio of the work required for an ideal (reversible) compression process to the actual work input. This assumes the compression process occurs between states 1 and 2 *for a given pressure ratio*. In most cases, the pressure ratio is the *total pressure at state 2* divided by the *total pressure at state 1*. If the process is also adiabatic, then the ideal state at 2 is the *isentropic* state.

From the foregoing definition, the efficiency for an adiabatic compression process can be written as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\eta_c = \frac{h_{t2,i} - h_{t1}}{h_{t2} - h_{t1}} \tag{31–18}$$

where,

h_{t1} = total enthalpy at 1

h_{t2} = actual total enthalpy at 2

$h_{t2,i}$ = isentropic total enthalpy at 2

If the specific heat is constant, *Equation 31–18* (p. 1493) can also be expressed as

$$\eta_c = \frac{T_{t2,i} - T_{t1}}{T_{t2} - T_{t1}} \tag{31–19}$$

where,

T_{t1} = total temperature at 1

T_{t2} = actual total temperature at 2

$T_{t2,i}$ = isentropic total temperature at 2

Using the isentropic relation

$$\frac{T_{t2,i}}{T_{t1}} = \left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}} \tag{31–20}$$

where γ is the ratio of specific heats specified in the *Reference Values Task Page* (p. 1898).

The efficiency can be written in the compact form

$$\eta_c = \frac{T_{t1}\left[\left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]}{T_{t2} - T_{t1}} \tag{31–21}$$

Note that this definition requires data only for the *actual* states 1 and 2.

Compressor designers also make use of the polytropic efficiency when comparing one compressor with another. The polytropic efficiency is defined as follows:

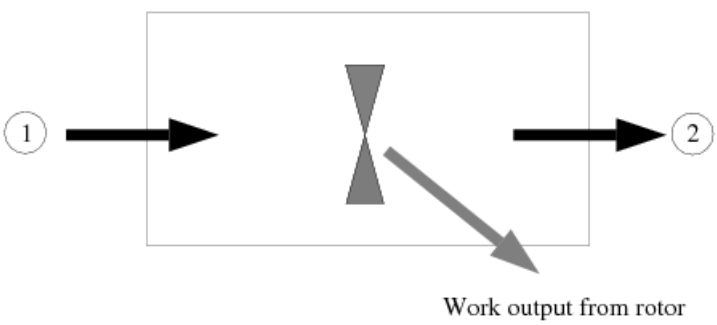
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\eta_{c,p} = \frac{\frac{\gamma-1}{\gamma}\ln\left(\frac{p_{t2}}{p_{t1}}\right)}{\ln\left(\frac{T_{t2}}{T_{t1}}\right)} \tag{31–22}$$

31.10.2.1.10. Efficiencies for Turbines

Consider a turbine operating between states 1 and 2 in *Figure 31.89* (p. 1494). Work is extracted from the working fluid as it expands through the turbine. Assuming that the processes are steady state, steady flow, and that the mass flow rates are equal at the inlet and outlet of the device (no film cooling, bleed air removal, etc.), turbine efficiencies for incompressible and compressible flows are as described below.

Figure 31.89 Turbine



31.10.2.1.10.1. Incompressible Flows

The efficiency of a turbine with an incompressible working fluid is defined as the ratio of the work delivered to the rotor to the energy available from the fluid stream. This ratio can be expressed as follows:

$$\eta = \frac{T\omega}{Q\left(p_{t1} - p_{t2}\right)} \tag{31–23}$$

where,

Q = volumetric flow rate

p_t = total pressure

T = net torque acting on the rotor/impeller

ω = rotational speed

Note the similarity between this definition and the definition of incompressible compression efficiency (*Equation 31–17* (p. 1492)). As with hydraulic pumps and compressors, other efficiencies (e.g., volumetric and mechanical efficiencies) can be defined to account for other losses in the system.

31.10.2.1.10.2. Compressible Flows

For high-speed gas turbines operating at large expansion pressure ratios, compressibility must be accounted for. The efficiency of an expansion process with a compressible working fluid is defined as the ratio of the actual work extracted from the fluid to the work extracted from an ideal (reversible) process. This assumes that the expansion process occurs between states 1 and 2 *for a given pressure ratio*. In contrast to the com-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pression process, the pressure ratio for expansion is the *total pressure at state 1* divided by the *total pressure at state 2*. If the process is also adiabatic, then the ideal state at 2 is the *isentropic* state.

From the foregoing definition, the efficiency for an adiabatic expansion process through a turbine can be written as

$$\eta_c = \frac{h_{t1} - h_{t2}}{h_{t1} - h_{t2,i}} \tag{31–24}$$

where

h_{t1} = total enthalpy at 1

h_{t2} = actual total enthalpy at 2

$h_{t2,i}$ = isentropic total enthalpy at 2

If the specific heat is constant, *Equation 31–24* (p. 1495) can also be expressed as

$$\eta_e = \frac{T_{t1} - T_{t2}}{T_{t1} - T_{t2,i}} \tag{31–25}$$

where

T_{t1} = total temperature at 1

T_{t2} = actual total temperature at 2

$T_{t2,i}$ = isentropic total temperature at 2

Using the isentropic relation

$$\frac{T_{t1}}{T_{t2,i}} = \left(\frac{p_{t1}}{p_{t2}}\right)^{\frac{\gamma-1}{\gamma}} \tag{31–26}$$

the expansion efficiency can be written in the compact form

$$\eta_e = \frac{T_{t1} - T_{t2}}{T_{t1}\left[1 - \left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}}\right]} \tag{31–27}$$

Note that this definition requires data only for the *actual* states 1 and 2.

As with compressors, one may also define a polytropic efficiency for turbines. The polytropic efficiency is defined as follows:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\eta_{e,p} = \frac{\ln\left(\frac{T_{t1}}{T_{t2}}\right)}{\frac{\gamma-1}{\gamma}\ln\left(\frac{p_{t1}}{p_{t2}}\right)} \tag{31–28}$$

31.10.3. Displaying Turbomachinery Averaged Contours

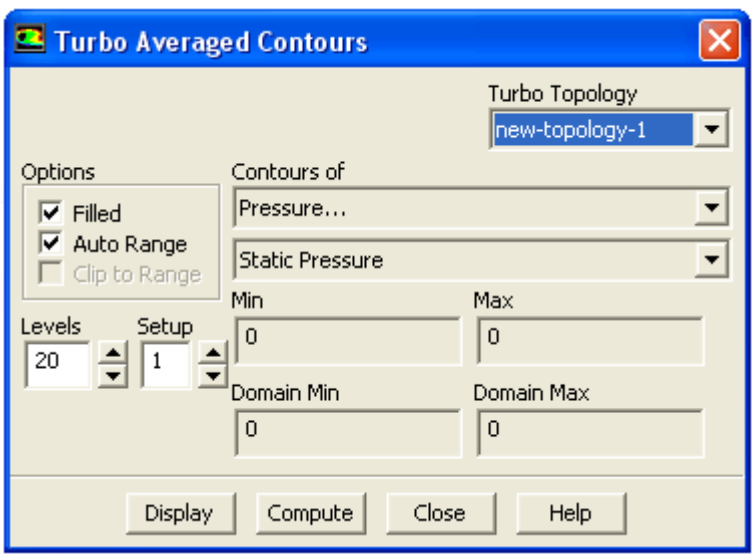
Turbo averaged contours are generated as projections of the values of a variable averaged in the circumferential direction and visualized on an r- z plane. A sample plot is shown in *Figure 31.91* (p. 1497).

31.10.3.1. Steps for Generating Turbomachinery Averaged Contour Plots

You can display contours using the *Turbo Averaged Contours Dialog Box* (p. 2175) (*Figure 31.90* (p. 1496)).

Turbo → **Averaged Contours...**

Figure 31.90 The Turbo Averaged Contours Dialog Box



The basic steps for generating a turbo averaged contour plot are as follows:

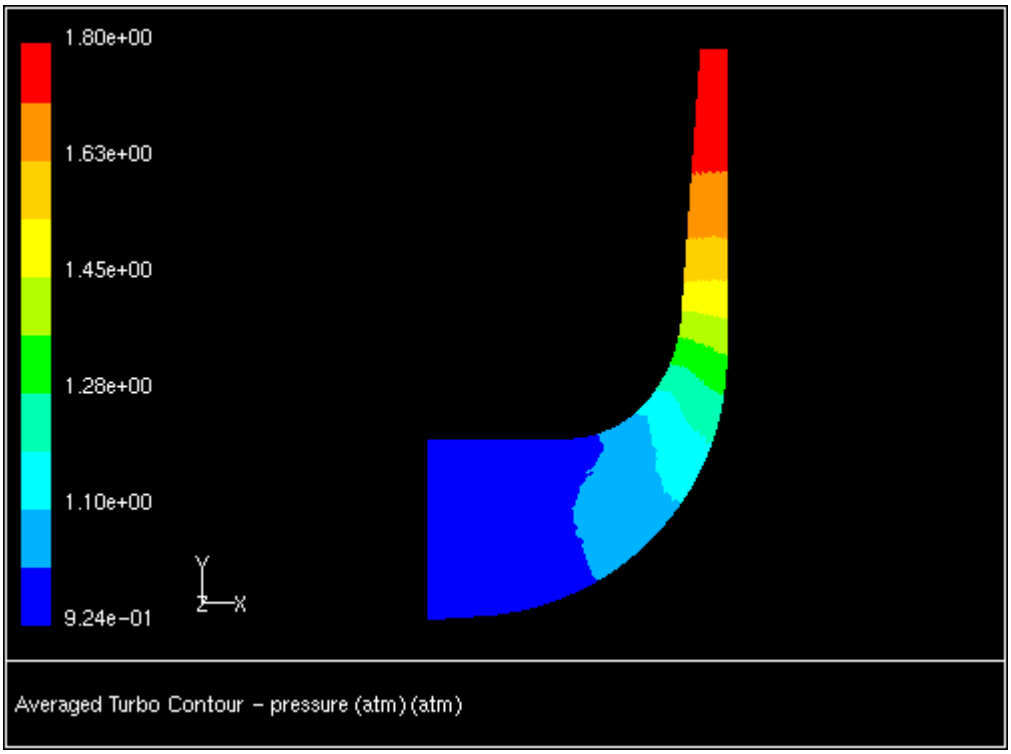
1. Select **All** or a specific predefined turbomachinery topology from the **Turbo Topology** drop-down list.
2. Select the variable or function to be displayed in the **Contours of** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See *Turbomachinery-Specific Variables* (p. 1500) for a list of turbomachinery-specific variables, and see *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
3. Specify the number of contours in the **Levels** field. The maximum number of levels allowed is 100.
4. Set any of the options described below.
5. Click the **Display** button to draw the specified contours in the active graphics window.

The resulting display will include the specified number of contours of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Min** and **Max** values displayed in the dialog box are the minimum and maximum averaged values. These limits will in general be different from the global **Domain Min** and **Domain Max**, which are also displayed for your reference (see *Figure 31.90* (p. 1496)).

Figure 31.91 Turbo Averaged Filled Contours of Static Pressure



31.10.3.2. Contour Plot Options

The options mentioned in the procedure above include drawing color-filled contours (instead of line contours), specifying a range of values to be contoured, and storing the contour plot settings. These options are the same as those in the standard *Contours Dialog Box* (p. 1967). See *Contour and Profile Plot Options* (p. 1389) for details about using them.

31.10.4. Displaying Turbomachinery 2D Contours

In postprocessing a turbomachinery solution, it is often desirable to display contours on surfaces of constant spanwise coordinate, and then project these contours onto a plane. This permits easier evaluation of the contours, especially for surfaces that are highly three-dimensional.

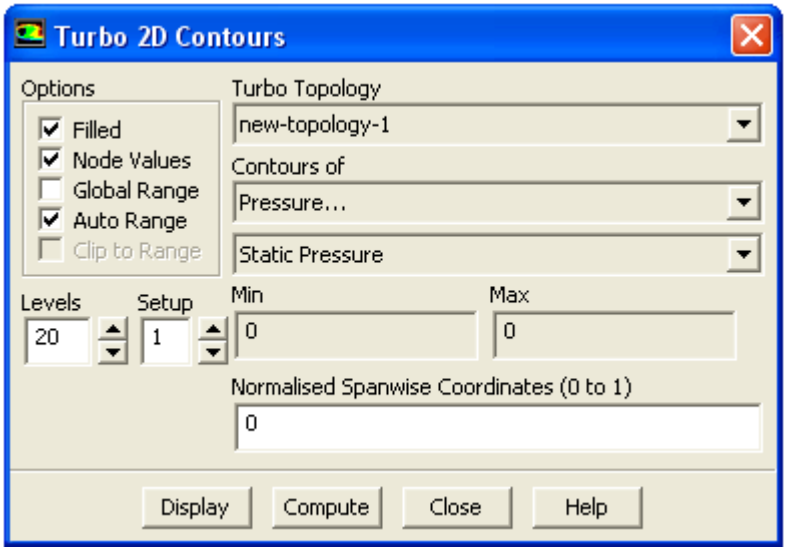
31.10.4.1. Steps for Generating Turbo 2D Contour Plots

You can display contours using the *Turbo 2D Contours Dialog Box* (p. 2176) (*Figure 31.92* (p. 1498)).

Turbo → **2D Contours...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.92 The Turbo 2D Contours Dialog Box



The basic steps for generating a turbo 2D contour plot are as follows:

1. Specify a specific predefined turbomachinery topology using the **Turbo Topology** drop-down list.
2. Enter a value for the **Normalized Spanwise Coordinates (0 to 1)** for the spanwise surface you want to create.

Important

If shroud and hub are the curved surfaces, the iso-surface very close to them may contain void spaces as ANSYS FLUENT displays only a plane cut surface.

---

3. Select the variable or function to be displayed in the **Contours of** drop-down list.

   First select the desired category in the upper list; you may then select a related quantity in the lower list. See *Turbomachinery-Specific Variables* (p. 1500) for a list of turbomachinery-specific variables, and see *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.
4. Specify the number of contours in the **Levels** field. The maximum number of levels allowed is 100.
5. Click the **Display** button to draw the specified contours in the active graphics window.

The resulting display will include the specified number of contours of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

31.10.4.2. Contour Plot Options

Depending on the type of contour plot you want to display, select appropriate choice under **Options**. These options are the same as those in the standard *Contours Dialog Box* (p. 1967). See *Contour and Profile Plot Options* (p. 1389) for details about using them.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.5. Generating Averaged XY Plots of Turbomachinery Solution Data

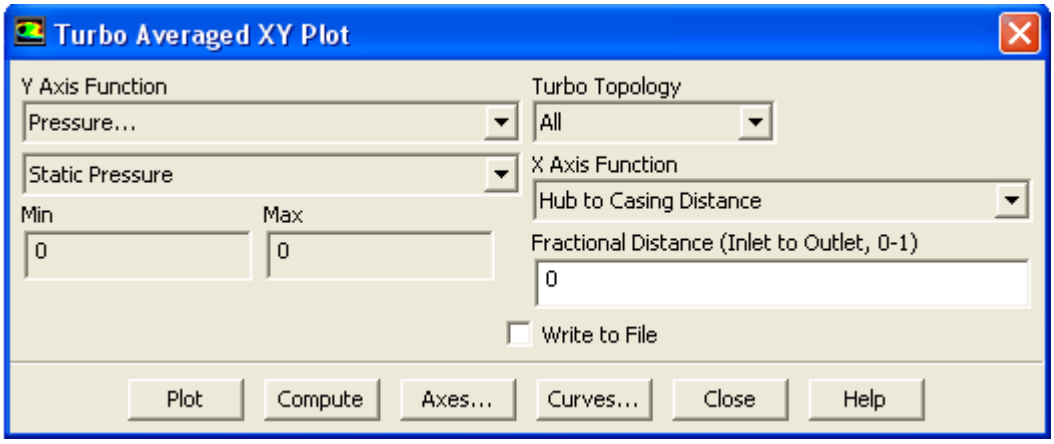
When comparing numerical solutions of turbomachinery problems to experimental data, it is often useful to plot circumferentially averaged quantities in the spanwise and meridional directions. This section describes how to do this in ANSYS FLUENT.

31.10.5.1. Steps for Generating Turbo Averaged XY Plots

To create an XY plot of circumferentially averaged solution data, you will use the *Turbo Averaged XY Plot Dialog Box* (p. 2178) (*Figure 31.93* (p. 1499)).

Turbo → **Averaged XY Plot...**

Figure 31.93 The Turbo Averaged XY Plot Dialog Box



The procedure for generating a turbo averaged XY plot are as follows:

1. Select the variable or function to be plotted in the **Y Axis Function** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See *Turbomachinery-Specific Variables* (p. 1500) for a list of turbomachinery-specific variables, and see *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
2. Select **All** or a specific predefined turbomachinery topology from the **Turbo Topology** drop-down list.
3. Select the variable or function to be plotted in the **X Axis Function** drop-down list. The choices are **Hub to Casing Distance** and **Meridional Distance**.
4. Specify the desired value in the **Fractional Distance** field. The definition of the fractional distance depends on your selection of **X Axis Function**:
   - If you select **Hub to Casing Distance**, the fractional distance will be **Inlet to Outlet**.
   - If you select **Meridional Distance**, the fractional distance will be **Hub to Casing**.
5. (optional) Modify the attributes of the axes or curves as described in *Modifying Axis Attributes* (p. 1480) and *Modifying Curve Attributes* (p. 1482).
6. Click the **Plot** button to generate the XY plot in the active graphics window.

   Note that you can use any of the mouse buttons to annotate the XY plot (see *Adding Text to the Graphics Window* (p. 1415)).

If you wish to write the XY data to a file, follow these steps instead of Step 5 above:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

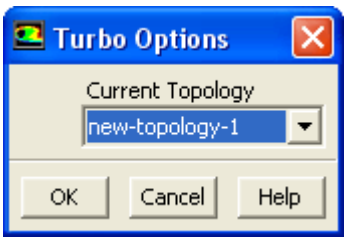
1. Enable the **Write to File** option. The **Plot** button will change to the **Write...** button.
2. Click **Write...**.
3. In *The **Select File** Dialog Box* (p. 33), specify a name for the plot file and save it.

31.10.6. Globally Setting the Turbomachinery Topology

In some cases, i.e., iso-surface creation, ANSYS FLUENT allows you to globally set the current turbomachinery topology for your model using the *Turbo Options Dialog Box* (p. 2179) (*Figure 31.94* (p. 1500)).

Turbo → **Options...**

Figure 31.94 The Turbo Options Dialog Box



To set the current topology, select a topology from the **Current Topology** drop-down list and select **OK**.

31.10.7. Turbomachinery-Specific Variables

The following turbomachinery-specific variables are available in ANSYS FLUENT:

- **Meridional Coordinate**
- **Abs Meridional Coordinate**
- **Spanwise Coordinate**
- **Abs (H-C) Spanwise Coordinate**
- **Abs (C-H) Spanwise Coordinate**
- **Pitchwise Coordinate**
- **Abs Pitchwise Coordinate**

These variables are contained in the **Mesh...** category of the variable selection drop-down list. See *Field Function Definitions* (p. 1529) for their definitions.

31.11. Fast Fourier Transform (FFT) Postprocessing

When trying to interpret time-sequence data from a transient solution, it is often useful to look at the data's spectral (frequency) attributes. For instance, you may wish to determine the major vortex-shedding frequency from the time-history of the drag force on a body recorded during an ANSYS FLUENT simulation. Or, you may want to compute the spectral distribution of static pressure data recorded at a particular location on a body surface. Similarly, you may need to compute the spectral distribution of turbulent kinetic energy using data for fluctuating velocity components. To interpret some of these time dependent data, you need to perform Fourier transform analysis. In essence, the Fourier transform enables you to take any time dependent data and resolve it into an equivalent summation of sine and cosine waves.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT allows you to analyze your time dependent data using the Fast Fourier Transform (FFT) algorithm. Information on using the FFT algorithm in ANSYS FLUENT is provided in the following sections:

31.11.1. Limitations of the FFT Algorithm
31.11.2. Windowing
31.11.3. Fast Fourier Transform (FFT)
31.11.4. Using the FFT Utility

31.11.1. Limitations of the FFT Algorithm

The following limitations apply to ANSYS FLUENT's FFT module:

- The ANSYS FLUENT FFT module can only read inputs files in the ANSYS FLUENT monitor and x-y file formats.
- The ANSYS FLUENT FFT module assumes that the input data have been sampled at equal intervals and are consecutive (in the order of increasing time).
- The lowest frequency that the FFT module can pick up is given by $1/t$, where t is the total sampling time. If the sampled sequence contains frequencies lower than this, these frequencies will be aliased into higher frequencies.
- The highest frequency that the FFT module can pick up is $1/(2dt)$, where dt is the sampling interval (or time step).

31.11.2. Windowing

The discrete FFT algorithm is based on the assumption that the time-sequence data passed to the FFT corresponds to a single period of a periodically repeating signal. Since, in most situations, the first and the last data points will not coincide, the repeating signal implied in the assumption can often have a large discontinuity. The large discontinuity produces high-frequency components in the resulting Fourier modes, causing an aliasing error. You can condition the input signal before the transform by "windowing" it, in order to avoid this problem.

Suppose that we have N consecutive discrete (time-sequence) data sampled with a constant interval, Δt:

$$\phi_k \equiv \phi(t_k), \qquad t_k \equiv k\Delta t, \qquad k = 0, 1, 2, \ldots, (N-1) \tag{31–29}$$

Windowing is done by multiplying the original input data (ϕ_j) by a window function, W_j:

$$\tilde{\phi}_j = \phi_j W_j \qquad j = 0, 1, 2, \ldots, (N-1) \tag{31–30}$$

ANSYS FLUENT offers four different window functions:

Hamming's window:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$W_j = \begin{cases} 0.54 - 0.46\cos\left(\frac{8\pi j}{N}\right) & j \le \frac{N}{8}, j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31–31}$$

Hanning's window:

$$W_j = \begin{cases} 0.5\left[1 - \cos\left(\frac{8\pi j}{N}\right)\right] & j \le \frac{N}{8}, j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31–32}$$

Barlett's window:

$$W_j = \begin{cases} \frac{8j}{N} & j \le \frac{N}{8} \\ 8\left(1 - \frac{j}{N}\right) & j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31–33}$$

Blackman's window:

$$W_j = \begin{cases} 0.42 - 0.5\cos\left(\frac{8\pi j}{N}\right) + 0.08\cos\left(\frac{16\pi j}{N}\right) & j \le \frac{N}{8}, j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31–34}$$

These window functions preserve a large fraction $(3/4)$ of the original data, affecting only $1/4$ of the data on both ends.

31.11.3. Fast Fourier Transform (FFT)

The Fourier transform utility in ANSYS FLUENT allows you to compute the Fourier transform of a signal, $\phi(t)$, a real-valued function, from a finite number of its sampled points.

The discrete Fourier transform of ϕ_k is defined by

$$\phi_k = \sum_{n=0}^{N-1} \hat{\phi}_n e^{2\pi i k n / N} \qquad k = 0, 1, 2, \ldots (N-1) \tag{31–35}$$

where $\hat{\phi}_n$ are the discrete Fourier coefficients, which can be obtained from where $\hat{\phi}_n$ are the discrete Fourier coefficients, which can be obtained from

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\hat{\phi}_n = \frac{1}{N}\sum_{k=0}^{N-1}\phi_k e^{-2\pi ikn/N} \qquad n=0,1,2,\ldots(N-1) \tag{31–36}$$

Equation 31–35 (p. 1502) and *Equation 31–36* (p. 1503) form a Fourier transform pair that allows us to determine one from the other.

Note that when we follow the convention of varying n from 0 to $N-1$ in *Equation 31–35* (p. 1502) or *Equation 31–36* (p. 1503) instead of from $-N/2$ to $N/2$, the range of index $1 \le n \le N/2-1$ corresponds to positive frequencies, and the range of index $N/2+1 \le n \le N-1$ corresponds to negative frequencies. $n=0$ still corresponds to zero frequency.

For the actual calculation of the transforms, ANSYS FLUENT adopts the so-called fast Fourier transform (FFT) algorithm which significantly reduces operation counts in comparison to the direct transform. Furthermore, unlike most FFT algorithms in which the number of data should be a power of 2, the FFT utility in ANSYS FLUENT employs a prime-factor algorithm [*94*] (p. 2214). The number of data points permissible in the prime-factor FFT algorithm is any products of mutually prime factors from the set 2,3,4,5,7,8,9,11,13,16, with a maximum value of $720720 = 5\times7\times9\times11\times13\times16$. Thus, the prime-factor FFT preserves the original data better than the conventional FFT.

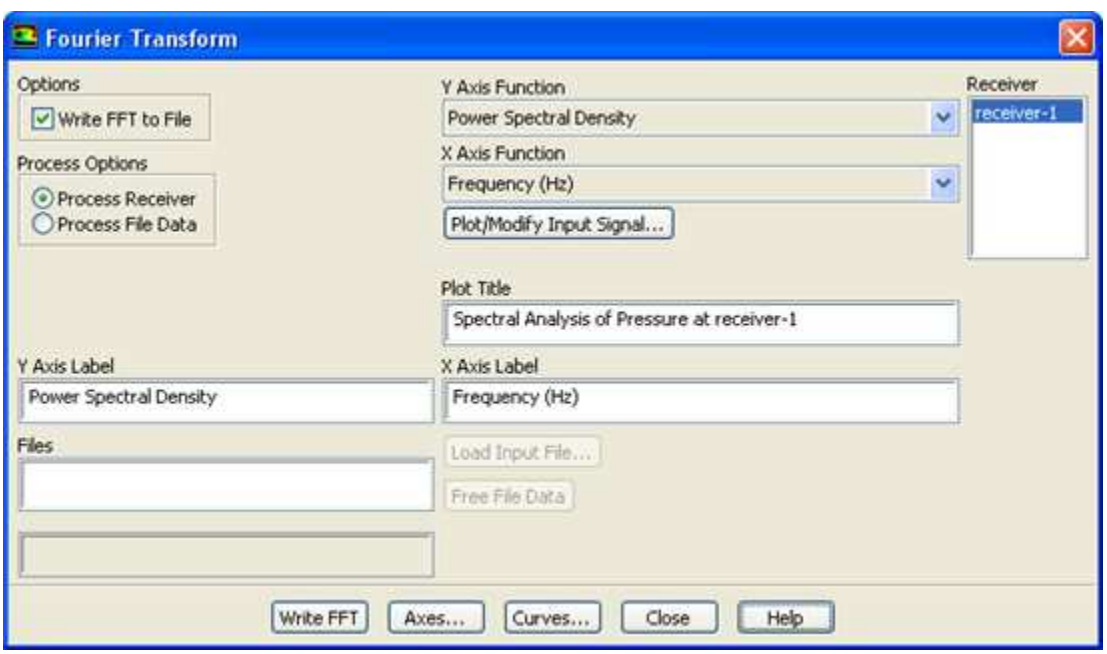
Just prior to computing the transform, ANSYS FLUENT determines the largest permissible number of data points based on the prime factors, discarding the rest of the data.

31.11.4. Using the FFT Utility

The ANSYS FLUENT FFT utility is available through the *Fourier Transform Dialog Box* (p. 2019) (*Figure 31.95* (p. 1503)).

Plots → **FFT** → **Set Up...**

Figure 31.95 The Fourier Transform Dialog Box



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.11.4.1. Loading Data for Spectral Analysis

FFT analysis requires an input signal data file consisting of time-sequence data. To load an input signal data file into the *Fourier Transform Dialog Box* (p. 2019), click the **Load Input File...** button. This displays *The **Select File** Dialog Box* (p. 33) where you can browse through your file directories and locate your data file containing your time-sequence data. To remove a file from the **Files** list, select it and then click the **Free File Data** button.

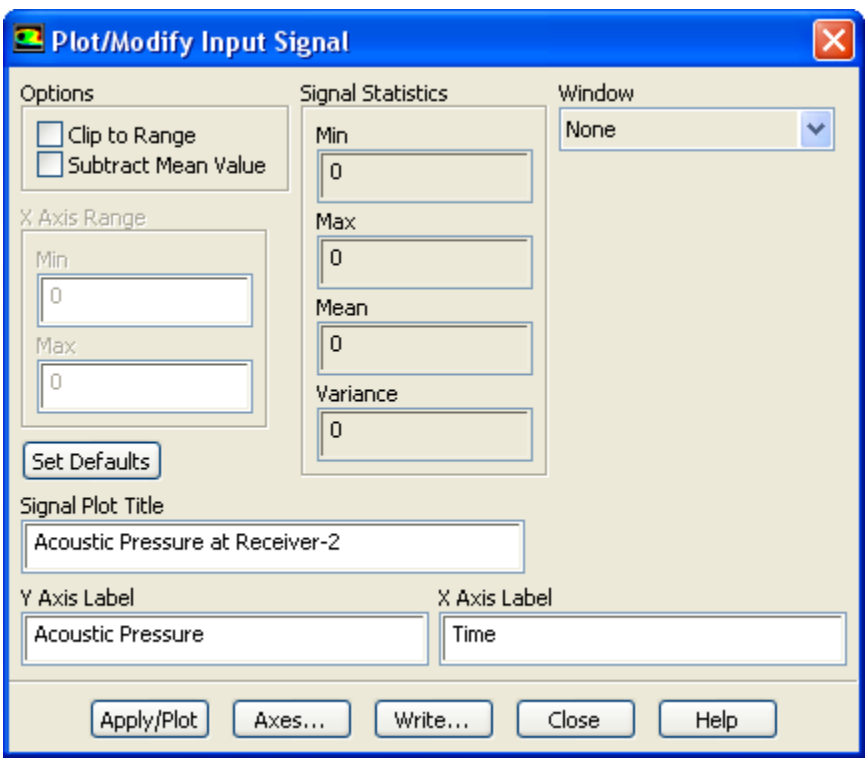
If you computed acoustic signals "on the fly", you have the option of processing signal data from a file or processing receiver data stored in memory. To analyze signal data from an existing input file, select **Process File Data** under **Process Options** and proceed as described above. To analyze receiver data stored in memory, select **Process Receiver** under **Process Options** and select the appropriate receiver in the **Receiver** list.

Click **Plot FFT** to display the spectral analysis data.

31.11.4.2. Customizing the Input

With the input signal data file loaded into the *Fourier Transform Dialog Box* (p. 2019), you may want to customize the input signal data set. You can customize the input signal by clicking the **Plot/Modify Input Signal** button. This displays the *Plot/Modify Input Signal Dialog Box* (p. 2020) (*Figure 31.96* (p. 1504)).

Figure 31.96 The Plot/Modify Input Signal Dialog Box



The *Plot/Modify Input Signal Dialog Box* (p. 2020) allows you to analyze a portion of the input signal, view input **Signal Statistics** (**Min**, **Max**, **Mean**, and **Variance**), and set title and label information for the input signal data file.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.11.4.2.1. Customizing the Input Signal Data Set

By default, the entire data set is analyzed. To analyze a portion of the input signal, enable the **Clip to Range** option and specify the data range by entering **Min** and **Max** values under **X-Axis Range**. To have the y axis quantities reduced by the **Mean** value of the relevant signal property, enable the **Subtract Mean Value** option.

The **Set Defaults** button will reset the original values for the **Min** and **Max** fields under **X-Axis Range** and turn off the **Clip to Range** option.

31.11.4.2.2. Viewing Data Statistics

To aid in the signal analysis, whether for the entire input signal or for a certain range of data, the **Signal Statistics** portion of the *Plot/Modify Input Signal Dialog Box* (p. 2020) displays signal information such as minimum, maximum, and average signal values, as well as signal variance.

31.11.4.2.3. Customizing Titles and Labels

You can create a new title or edit the original title for the input signal plot by entering a text string in the **Signal Plot Title** text box. Likewise, you can create a new axis label or edit the original axis label by enteringa text string into either the **Y-Axis Label** text box or the **X-Axis Label** text box.

31.11.4.2.4. Applying the Changes in the Input Signal Data

To apply any changes you have made in the *Plot/Modify Input Signal Dialog Box* (p. 2020) and view a plot of the input signal, click the **Apply/Plot** button.

31.11.4.3. Customizing the Output

In most practical applications with CFD data, you may want to find out how much power or energy is contained in a certain frequency range, but do not want to distinguish positive and negative frequency. In recognition of this, all the outputs from the FFT module in ANSYS FLUENT pertain to *one-sided spectra* for the range of positive frequency.

The *Fourier Transform Dialog Box* (p. 2019) (*Figure 31.95* (p. 1503)) and *Plot/Modify Input Signal Dialog Box* (p. 2020) (*Figure 31.96* (p. 1504)) allow you to set several different functions for the x and y axes, apply different FFT windowing techniques, and set various output options.

31.11.4.3.1. Specifying a Function for the y Axis

You can choose the y-axis function using the **Y Axis Function** drop-down list. Available options for the y-axis functions are as follows:

Power Spectral Density

is the distribution of signal power in the frequency domain. It has units of the signal magnitude squared (e.g., Pa^2) and is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$E(f_0) = \left|\hat{\phi}_0\right|^2$$
$$E(f_n) = 2\left|\hat{\phi}_n\right|^2 \qquad n=1,2,\ldots,N/2 \tag{31–37}$$

Magnitude

(or amplitude) is the square root of the power spectral density.

$$A(f_n) \equiv \sqrt{E(f_n)} \qquad n=0,1,2,\ldots,N/2 \tag{31–38}$$

Sound Pressure Level (dB)

is the decibel level. For either general or acoustic data, when the sampled data is pressure (e.g., static pressure or sound pressure), you can compute the power in decibel units using

$$L_{sp}(f_n) = 10\log\left(\frac{p'^2(f_n)}{p_{ref}^2}\right) \qquad (dB) \tag{31–39}$$

where $p'^2(f_n)$ is the power spectral density of the pressure fluctuation and p_{ref} is the reference acoustic pressure. See *Enabling the FW-H Acoustics Model* (p. 990) for details about specifying this parameter.

Sound Amplitude (dB)

is exactly one-half of the sound pressure level in *Equation 31–39* (p. 1506). This quantity is also applicable for acoustics analysis.

$$A_{sp}(f_n) = 10\log\sqrt{\frac{p'^2(f_n)}{p_{ref}^2}} \qquad (dB) \tag{31–40}$$

A-Weighted, Sound Pressure Level (dB A)

is the calculated sound pressure level weighted by the A-scale function to more closely approximate the frequency response of the human ear. A-Weighting is applied for loudness levels below 55 phons (55 dB at 1 kHz) and is the most commonly used weighting function. See *Figure 31.97* (p. 1507) for a graphical representation.

B-Weighted, Sound Pressure Level (dB B)

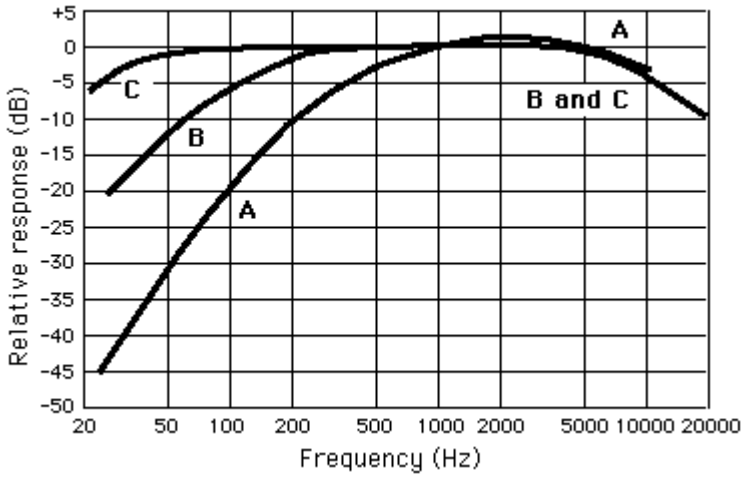
is the calculated sound pressure level weighted by the B-scale function. B-Weighting is applied to loudness levels between 55 and 85 phons, though it is rarely used.

C-Weighted, Sound Pressure Level (dB C)

is the calculated sound pressure level weighted by the C-scale function. C-Weighting is applied for loudness levels above 85 phons and is commonly used for high-intensity sound such as traffic studies.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.97 A-, B-, and C-Weighting Functions



Further graphical customizations for the y axis are available by clicking the **Axes...** button. For more information, see *Modifying Axis Attributes* (p. 1480).

31.11.4.3.2. Specifying a Function for the x Axis

There are three options for the x-axis function you can choose from. They are all related to the discrete frequencies at which the Fourier coefficients are computed. You can apply specific analytic functions for the x-axis using the **X Axis Function** drop-down list.

Available options for the x-axis functions are:

Frequency (Hz)
is defined as:

$$f_n = \frac{1}{N\Delta t} n \qquad n = 0, 1, 2, \ldots, N/2 \tag{31–41}$$

where N is the number of data points used in the FFT.

Strouhal Number
is the nondimensionalized version of the frequency defined in *Equation 31–41* (p. 1507) :

$$St_n \equiv \frac{f_n L_{ref}}{U_{ref}} \tag{31–42}$$

where L_{ref} and U_{ref} are the reference length and velocity scales specified in the *Reference Values Task Page* (p. 1898).

Fourier Mode
is the index in *Equation 31–35* (p. 1502) and/or *Equation 31–36* (p. 1503), which represents the nth or kth term in the Fourier transform of the signal.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Octave Band (Hz)
is a range of discrete frequency bands for different octaves within the threshold of hearing. The range of each octave band is double to that of the previous band (see *Table 31.2: Octave Band Frequencies and Weightings* (p. 1508)).

1/3-Octave Band (Hz)
is a range of discrete frequency bands within the threshold of hearing.

Table 31.2 Octave Band Frequencies and Weightings

Lower Freq. (Hz)	Center Freq. (Hz)	Upper Freq. (Hz)	dB A	dB B	dB C
11	16	22	-56.7	-28.5	-8.5
22	31.5	45	-39.4	-17.1	-3.0
45	63	90	-26.2	-9.3	-0.8
90	125	180	-16.1	-4.2	-0.2
180	250	355	-8.6	-1.3	0.0
355	500	710	-3.2	-0.3	0.0
710	1000	1400	0.0	0.0	0.0
1400	2000	2800	1.2	-0.1	-0.2
2800	4000	5600	1.0	-0.7	-0.8
5600	8000	11200	-1.1	-2.9	-3.0
11200	16000	22400	-6.6	-8.4	-8.5

Further graphical customizations for the x-axis are available by clicking the **Axes...** button. For more information, see *Modifying Axis Attributes* (p. 1480).

31.11.4.3.3. Specifying Output Options

You can write out the FFT data directly to a file by choosing the **Write FFT to File** option under **Options** in the *Fourier Transform Dialog Box* (p. 2019). Once the **Write FFT to File** option is selected, click the **Write FFT** button to display a file selection dialog box where you can choose a file and/or a location to hold the FFT data.

Further customizations for how the FFT data is displayed are available by clicking the **Curves...** button. For more information, see *Modifying Curve Attributes* (p. 1482).

31.11.4.3.4. Specifying a Windowing Technique

You can use the various windowing techniques described in *Windowing* (p. 1501) by selecting any of the **Window** options in the *Plot/Modify Input Signal Dialog Box* (p. 2020). By default, **None** is selected so that no windowing technique is applied.

31.11.4.3.5. Specifying Labels and Titles

You can assign a title for your FFT plot using the **Plot Title** text field. You can also assign y-axis and x-axis labels for your FFT plot using the **Y-Axis Label** and **X-Axis Label** text fields, respectively. By default,ANSYS FLUENT assigns the **Y-Axis Label** and the **X-Axis Label** to the particular selection of **Y-Axis Function** and **X-Axis Function**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 32: Reporting Alphanumeric Data

ANSYS FLUENT provides tools for computing and reporting integral quantities at surfaces and boundaries. These tools enable you to find the mass flow rate and heat transfer rate through boundaries, the forces and moments on boundaries, and the area, integral, flow rate, average, and mass average (among other quantities) on a surface or in a volume. In addition, you can print histograms of geometric and solution data, set reference values for the calculation of non-dimensional coefficients, and compute projected surface areas. You can also print or save a summary report of the models, boundary conditions, and solver settings in the current case. These features are described in the following sections.

32.1. Reporting Conventions
32.2. Creating Output Parameters
32.3. Fluxes Through Boundaries
32.4. Forces on Boundaries
32.5. Projected Surface Area Calculations
32.6. Surface Integration
32.7. Volume Integration
32.8. Histogram Reports
32.9. Discrete Phase
32.10. S2S Information
32.11. Reference Values
32.12. Summary Reports of Case Settings
32.13. Memory and CPU Usage

Reporting tools for the discrete phase are described in *Postprocessing for the Discrete Phase* (p. 1067).

32.1. Reporting Conventions

For 2D problems, ANSYS FLUENT computes all integral quantities for a unit depth equivalent to 1 meter. This value can by adjusted to match the specific dimension of your application only by manually revising the **Depth** in the *Reference Values Task Page* (p. 1898) (see *Reference Values* (p. 1523)).

Important

The default value of **Depth** will be equivalent to 1 meter, even if the units are changed for **depth** in the *Set Units Dialog Box* (p. 1637) (e.g., if the units for **depth** are changed to cm in the *Set Units Dialog Box* (p. 1637), the value of **Depth** in the *Reference Values Task Page* (p. 1898) will be 100 cm).

For axisymmetric problems, all integral quantities are computed for an angle of 2π radians.

32.2. Creating Output Parameters

You can create output parameters, which allow you to compare reporting values for different cases. These are single values generated by the four types of reports:

- Fluxes (*Fluxes Through Boundaries* (p. 1510))
- Forces (*Forces on Boundaries* (p. 1515))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Surface integrals (*Generating a Surface Integral Report* (p. 1519))
- Volume integrals (*Generating a Volume Integral Report* (p. 1521))

In the *Reports Task Page* (p. 2026), click the **Parameters...** button to open the *Parameters Dialog Box* (p. 2041), where a list of created input parameters will be listed. The list of **Input Parameters** is available after performing the steps outlined in *Defining and Viewing Parameters* (p. 210). The list of **Output Parameters** is available after clicking the **Save Output Parameters...** button in the **Fluxes**, **Forces**, **Surface Integrals**, and **Volume Integrals** dialog boxes.

You can define the output parameters either through the various reporting dialog boxes, as described in the sections that follow, or through the **Create** menu. In the **Create** menu, you will find a list of four items:

- **Fluxes...**
- **Forces...**
- **Surface Integrals...**
- **Volume Integrals...**

Selecting any one of these items will open their respective dialog boxes, where you will define the type of report you would like to generate. Details on how to generate the various reports are available in *Fluxes Through Boundaries* (p. 1510), *Forces on Boundaries* (p. 1515), *Generating a Surface Integral Report* (p. 1519), and *Generating a Volume Integral Report* (p. 1521).

Once you have saved your output parameters, you can modify them by selecting the parameter in the **Output Parameters** list and clicking **Edit...**. This will open the report dialog box where you can make your changes.

In addition, you can select any of the following under the **More** menu:

Delete
: displays a message in a dialog box, prompting you for a response to confirm the deletion of the output parameter.

Rename
: allows you to edit the name of the output parameter.

Print to Console
: reports values to the console window. If you select multiple output parameters.

Print All to Console
: outputs the values from all output parameters to the console window.

Write...
: allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Write All...
: prompts you for a file name and then writes the values for all of the output parameters to a file.

32.3. Fluxes Through Boundaries

This section contains information about generating a flux report. For more background information, see Fluxes Through Boundaries in the Theory Guide.

For additional information, please see the following sections:

32.3.1. Generating a Flux Report
32.3.2. Flux Reporting for Reacting Flows

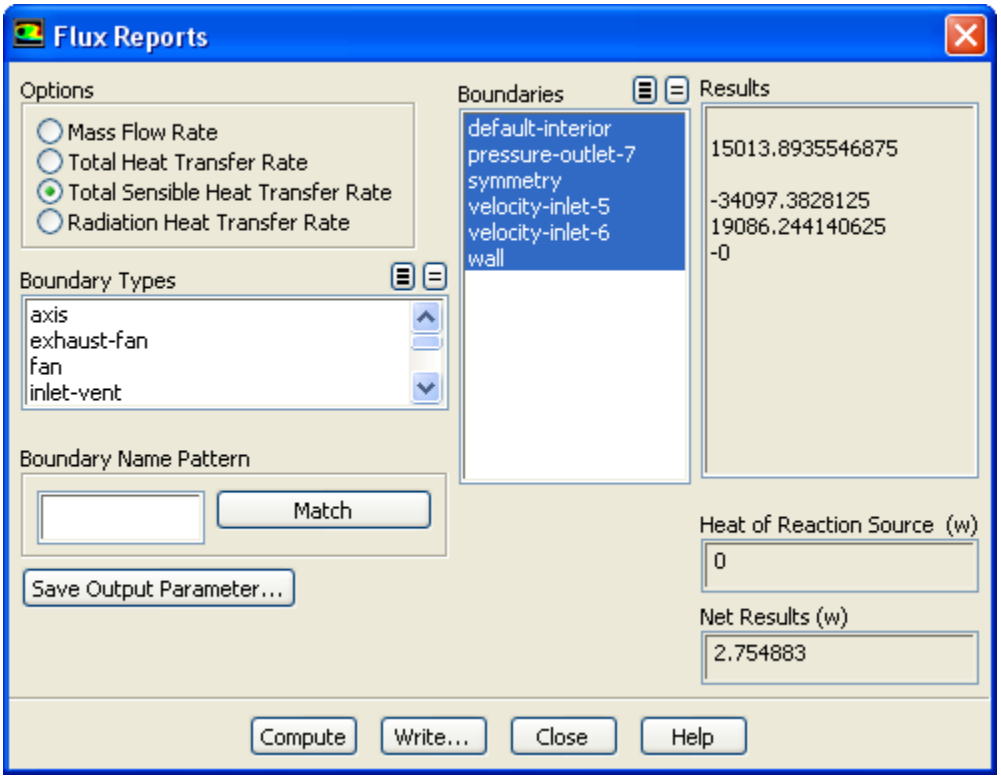
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.3.1. Generating a Flux Report

To obtain a report of mass flow rate, total heat transfer rate, total sensible heat transfer rate, or radiation heat transfer rate on selected boundary zones, use the *Flux Reports Dialog Box* (p. 2027) (*Figure 32.1* (p. 1511)).

Reports → **Fluxes** → **Set Up...**

Figure 32.1 The Flux Reports Dialog Box



The steps for generating the report are as follows:

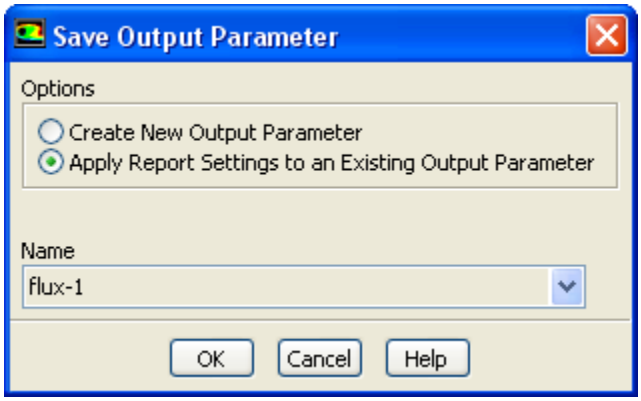
1. Specify which flux computation you are interested in by selecting **Mass Flow Rate**, **Total Heat Transfer Rate**, **Total Sensible Heat Transfer Rate**, or **Radiation Heat Transfer Rate** under **Options**.
2. In the **Boundaries** list, choose the boundary zone(s) on which you want to report fluxes.

   If you want to select several boundary zones of the same type, you can select that type in the **Boundary Types** list instead. All of the boundaries of that type will be selected automatically in the **Boundaries** list (or deselected, if they are all already selected).

   Another shortcut is to specify a **Boundary Name Pattern** and click **Match** to select boundary zones with names that match the specified pattern. For example, if you specify `wall*`, all boundaries whose names begin with **wall** (e.g., **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all boundaries whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all already selected).
3. Click **Save Output Parameter...**. The *Save Output Parameter Dialog Box* (p. 2044) (*Figure 32.2* (p. 1512)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. The default report name format is report-type-n (e.g. flux-1).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.2 The Save Output Parameter Dialog Box



After the output parameter is created, it is listed in the *Parameters Dialog Box* (p. 2041), accessed via the **Parameters...** button in the *Reports Task Page* (p. 2026). You can create any number of output parameters of this report type.

4. Click the **Compute** button to display the results of the selected flux computation for each selected boundary zone. The **Net Results** field will show the summation of the individual zone flux results.

Important

Additional steps must be taken prior to generating a flux report for an interior boundary zone that has the same fluid defined on either side. In such a case, the area vectors of the cell faces associated with the zone may have been automatically defined in an inconsistent manner when the mesh file was read into the solver. Since the flux for each individual cell face is calculated with respect to its area vector, such an inconsistency leads to inaccurate results when the face fluxes are summed to calculate the total flux of the boundary zone.

To ensure accurate flux results for such an interior zone, you must orient the area vectors by changing the definition of the zone **Type** to **wall**. You should then change the **Type** back to **interior** and proceed to generate the flux report.

Note that the fluxes are reported exactly as computed by the solver. Therefore, they are inherently more accurate than those computed with the **Flow Rate** option in the *Surface Integrals Dialog Box* (p. 2031) (described in *Surface Integration* (p. 1519)).

32.3.2. Flux Reporting for Reacting Flows

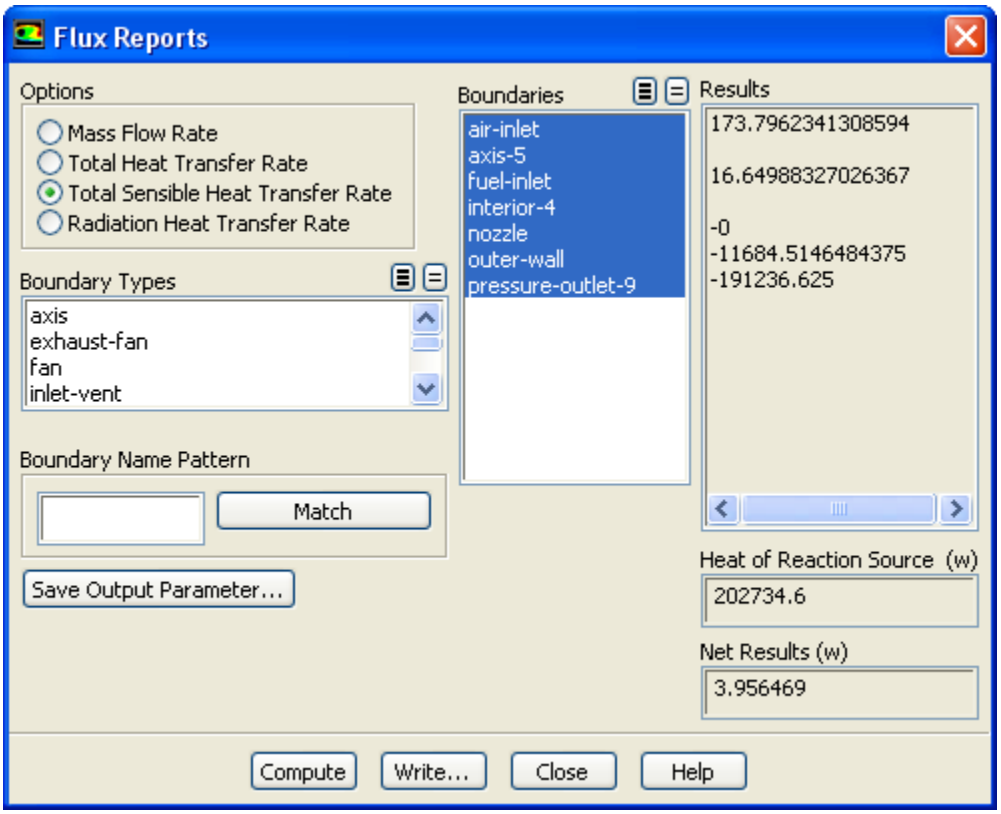
To report heat transfer for reacting flows, one of models in the *Species Model Dialog Box* (p. 1681) must be enabled for the **Total Sensible Heat Transfer Rate** option to appear in the *Flux Reports Dialog Box* (p. 2027). For reacting flows, ANSYS FLUENT produces two kinds of reports which use a different treatment at the flow boundaries:

- **Total Heat Transfer Rate** reports the total enthalpy flux, which consists of the total energy plus the species formation enthalpies. The heat rate based on this definition is a conserved quantity in reacting flows. See Heat Transfer Theory in the Theory Guide for details.
- **Total Sensible Heat Transfer Rate** reports the total energy flux as defined in Equation 5–2 in the Theory Guide. Note that in reacting flows, this is not a conserved quantity and the addition or removal of heat due to the chemical reactions (Equation 5–10 in the Theory Guide) is reported separately in the **Heat of Reaction Source** field, as shown in *Figure 32.3* (p. 1513). If you have more than one reaction

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined in your case, the **Heat of Reaction Source** reported is the sum of the heat for all reactions. For exothermic reactions the **Heat of Reaction Source** is reported as a positive quantity, while for endothermic reactions it will be a negative quantity.

Figure 32.3 The Flux Reports Dialog Box



Important

Note that both the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options report a **Net Result**, which may be used as an indication of the energy balance for the case. In general, and if heat sources other than the heat of reaction and DPM are not included in your problem, the **Net Result** reported in both the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options should be a small number for a converged calculation. However, if a reacting case is not well converged for both energy and species transport equations, the **Net Result** reported in the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options may differ. In that case, you may consider iterating further to achieve a fully converged solution. In addition, please refer to the sections that follow for special considerations when including particles, multiphase models, or other volumetric energy sources.

Important

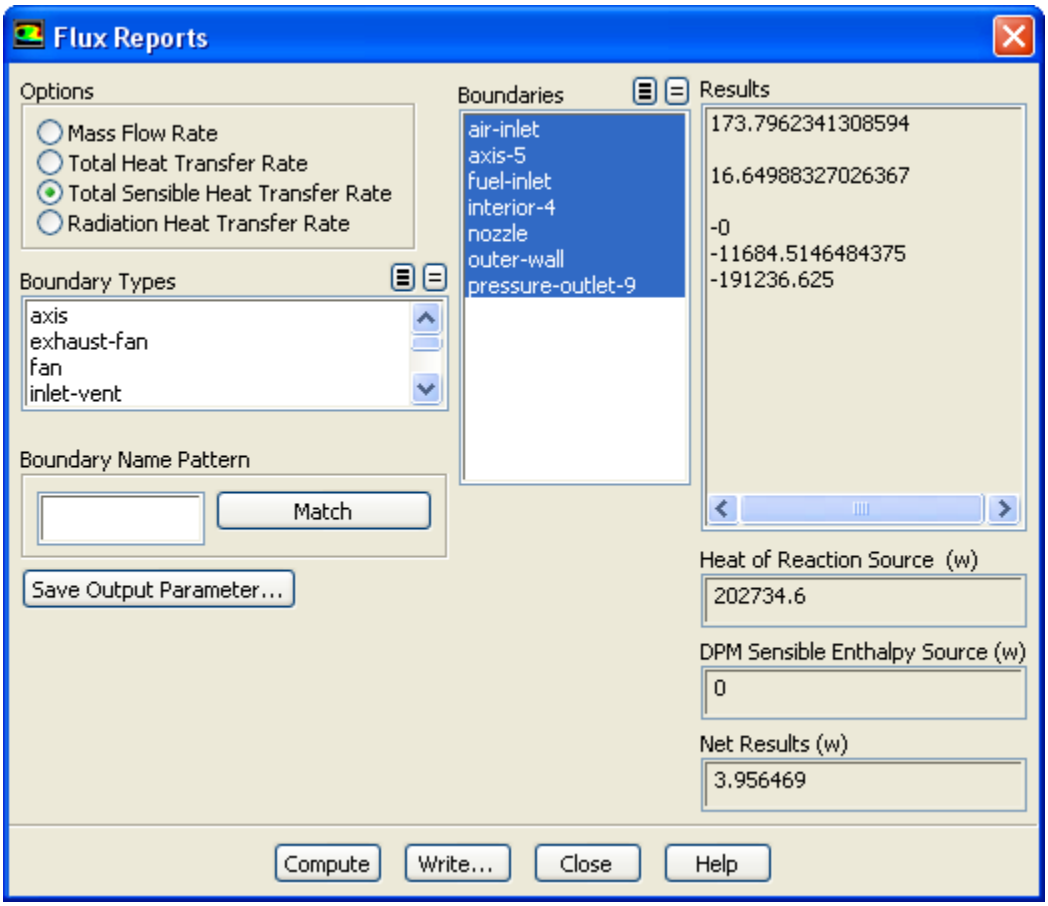
Please note that for the non-premixed and partially premixed models the **Heat of Reaction Source** is calculated as the difference of the net **Total Heat Transfer Rate** and the net **Total Sensible Heat Transfer Rate**. The **Heat of Reaction** field function is not available for the non-premixed and partially-premixed models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.3.2.1. Flux Reporting with Particles

If you are using the discrete phase model (DPM), the contributions from the particle injections are reported separately and are included in the net mass and heat balance results. Consequently, the **Mass Flow Rate** report includes the **DPM Mass Source**, the **Total Heat Transfer Rate** report includes the **DPM Enthalpy Source**, and the **Total Sensible Heat Transfer Rate** includes the **DPM Sensible Enthalpy Source** (*Figure 32.4* (p. 1514)).

Figure 32.4 The Flux Reports Dialog Box with DPM



Important

In the case of reacting flows with the DPM model, the **Heat of Reaction Source** entry reports the heat of all homogeneous reactions in the continuous phase, while the heat released or consumed due to particle reactions (e.g. char combustion) is reported in the **DPM Sensible Enthalpy Source** field.

32.3.2.2. Flux Reporting with Multiphase

If you are using any of the multiphase models, the mass or heat rates can be reported separately for each phase and for the mixture phase. Please note that if your multiphase model includes mass or heat transfer processes between phases, the mass and heat transferred across the phases will be reported as an imbalance in the report of each phase. In order to check the overall balances for the multiphase cases you should select the mixture phase for your report. In that case, the report will include the sum of the fluxes and sources for all phases included in your model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, if you are solving a multiphase problem that includes chemical reactions, you should be aware of the following conventions when you are requesting a **Total Sensible Heat Transfer Rate** report:

- If you select one of the phases with gas phase chemical reactions, the **Heat of Reaction Source** will only include contributions from reactions in the particular phase.
- When you report the **Total Sensible Heat Transfer Rate** for the mixture phase, the **Heat of Reaction Source** entry will report the sum of the heat of reaction of all gas phase reactions in all phases plus the heat of any heterogeneous reactions that take place.

32.3.2.3. Flux Reporting with Other Volumetric Sources

The reported mass and heat balances address the flow that enters or leaves the domain through boundaries and the contributions from DPM sources; they do not include the contributions from user-defined and other volumetric sources, such as the heat exchanged in the Heat Exchanger Model. For this reason, a mass or heat imbalance may be reported. In that case, and in a converged calculation, the reported imbalance will be equal to the volumetric source.

32.4. Forces on Boundaries

For selected wall zones, you can compute and report the forces along a specified vector, the moments about a specified center and along a specified axis, and the coordinates of the center of pressure. This feature is useful for reporting, for instance, aerodynamic quantities such as lift, drag, and moment coefficients, as well as the center of pressure for an airfoil.

For additional information about forces, moments, and the center of pressure, see Computing Forces, Moments, and the Center of Pressure in the Theory Guide.

For additional information, please see the following section:
32.4.1. Generating a Force, Moment, or Center of Pressure Report

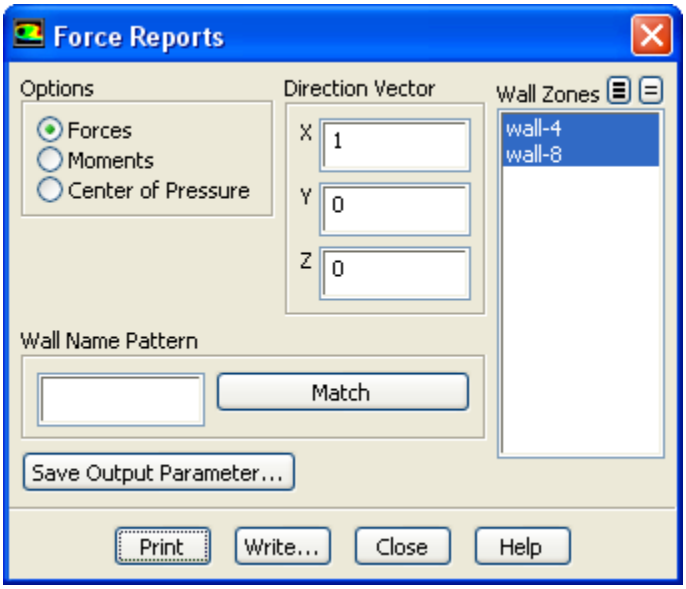
32.4.1. Generating a Force, Moment, or Center of Pressure Report

To obtain a report (for selected wall zones) of forces along a specified vector, moments about a specified center and along a specified axis, or the center of pressure, use the *Force Reports Dialog Box* (p. 2029) (*Figure 32.5* (p. 1516)).

Reports → **Forces** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.5 The Force Reports Dialog Box



The steps for generating the report are as follows:

1. Specify the type of report in which you are interested by selecting **Forces**, **Moments**, or **Center of Pressure** from the **Options** list.
2. Define the settings associated with report you are generating:
   a. For a force report, enter the **X**, **Y**, and **Z** components of the **Force Vector** along which the forces will be computed.
   b. For a moment report, enter the **X**, **Y**, and **Z** coordinates of the **Moment Center** about which the moments will be computed, as well as the **X**, **Y**, and **Z** components of the **Moment Axis** along which the moments will be computed.
   c. For a center of pressure report, define the line (for 2D geometries) or plane (for 3D geometries) on which you want to calculate the center of pressure. The line or plane must have one of its coordinate values fixed (e.g., a line defined as $y=10$). Select the axis (**X**, **Y**, or **Z**) in the **Coordinate** group box, and then enter the fixed **Value**. See the example at the end of this section for further details.

   Important

   If F_x (the x-component of the force) is zero, then either the **Y** or **Z** coordinate can be fixed. If F_y is zero, then either the **X** or **Z** coordinate can be fixed. If F_z is zero, then either the **X** or **Y** coordinate can be fixed.

3. In the **Wall Zones** list, select the wall zone(s) for which you want a report of the forces, moments, or pressure center.

   If you have a large number of wall zones, it may be useful to specify a **Wall Name Pattern** and click **Match**. This selects all of the wall zones with names that match the specified pattern. For example, if you specify `out*`, all walls whose names begin with **out** (e.g., **outer-wall-top**, **outside-wall**) will be selected automatically. If a wall zone that matches the name pattern is already selected when **Match**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is clicked, it will be deselected. If you specify `out?`, all walls whose names consist of **out** followed by a single character will be selected (or deselected, if they are already selected).

4. Click **Save Output Parameter...**. The *Save Output Parameter Dialog Box* (p. 2044) (*Figure 32.2* (p. 1512)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. The default report name format is report-type-n (e.g. force-1).

   After the output parameter is created, it is listed in the *Parameters Dialog Box* (p. 2041). You can create any number of output parameters of this report type.

5. Click the **Print** button if you want the results displayed in the console window, or click **Write...** to save it to a file.

   If you selected **Forces** under **Options**, the pressure force, viscous force (if appropriate), total forces, pressure coefficient, viscous coefficient, and total coefficients for each selected wall zone will be displayed or saved.

   If you selected **Moments**, the pressure moments, viscous moments (if appropriate), total moments, pressure coefficient, viscous coefficient and total coefficients for the wall zones about the specified center will be displayed or saved. Additionally, the moments and coefficients in the direction of the specified axis will be displayed or saved. The report will include the values for the individual wall zones, as well as the net values for all of the wall zones combined. See Computing Forces, Moments, and the Center of Pressure in the Theory Guide for details about computing forces and moments.

   If you selected **Center of Pressure**, then ANSYS FLUENT displays or saves the coordinates about which the moment is zero.

   Important

   You cannot save your output parameter if **Center of Pressure** is selected.

Important

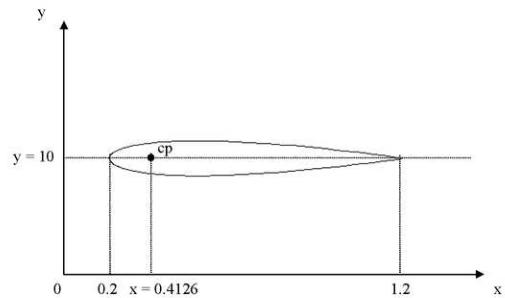
Note that the reported force and moment coefficients are a function of the values entered in the **Reference Values** task page (as described in Computing Forces, Moments, and the Center of Pressure in the Theory Guide). Therefore, appropriate values must be entered in the **Reference Values** task page to get meaningful results.

32.4.1.1. Example

To demonstrate how you would generate and interpret the center of pressure report, consider an airfoil of chord length 1 m (shown in *Figure 32.6* (p. 1518)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

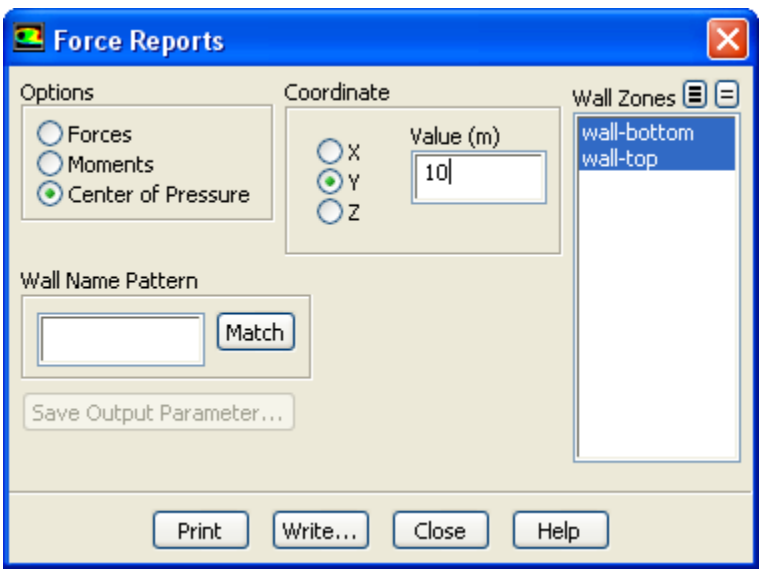
Figure 32.6 An Airfoil with its Computed Center of Pressure



Open the **Force Reports** dialog box and perform the steps that follow.

Reports → **Forces** → **Set Up...**

Figure 32.7 The Force Reports Dialog Box for a Center of Pressure Report



1. Select **Center of Pressure** from the **Options** list.
2. Define the line on which the center of pressure will be calculated. In this case, the **Y** coordinate for the line has a fixed **Value** of `10`.
3. Select the **Wall Zones** that are relevant for the computation.
4. Click **Print** to have the coordinates of the center of pressure displayed in the console window.

The report generated will be in the following form:

```
Pressure Center Coordinates (in m):
X = 0.41267981
Y = 10
```

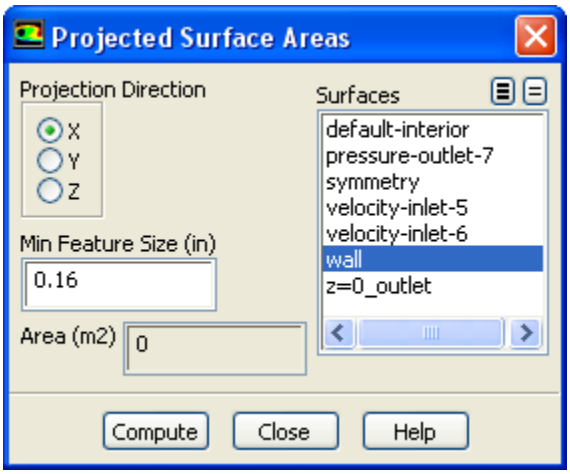
32.5. Projected Surface Area Calculations

You can use the *Projected Surface Areas Dialog Box* (p. 2030) (*Figure 32.8* (p. 1519)) to compute an estimated area of the projection of selected surfaces along the x, y, or z axis (i.e., onto the yz, xz, or xy plane).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reports → **Projected Areas** → **Set Up...**

Figure 32.8 The Projected Surface Areas Dialog Box



The steps for calculating the projected area are as follows:

1. Select the **Projection Direction** (**X**, **Y**, or **Z**).
2. Choose the surface(s) for which the projected area is to be calculated in the **Surfaces** list.
3. Set the **Min Feature Size** to the length of the smallest feature in the geometry that you want to resolve in the area calculation. (You can just use the default value to start with, if you are not sure of the size of the smallest geometrical feature.)
4. Click on **Compute**. The area will be displayed in the **Area** box and in the console window.
5. To improve the accuracy of the area calculation, reduce the **Min Feature Size** by half and recompute the area. Repeat this step until the computed **Area** stops changing (or you reach memory capacity).

This feature is available only for 3D domains.

32.6. Surface Integration

For additional information about surface integrals, see Computing Surface Integrals in the Theory Guide.

For additional information, please see the following section:
32.6.1. Generating a Surface Integral Report

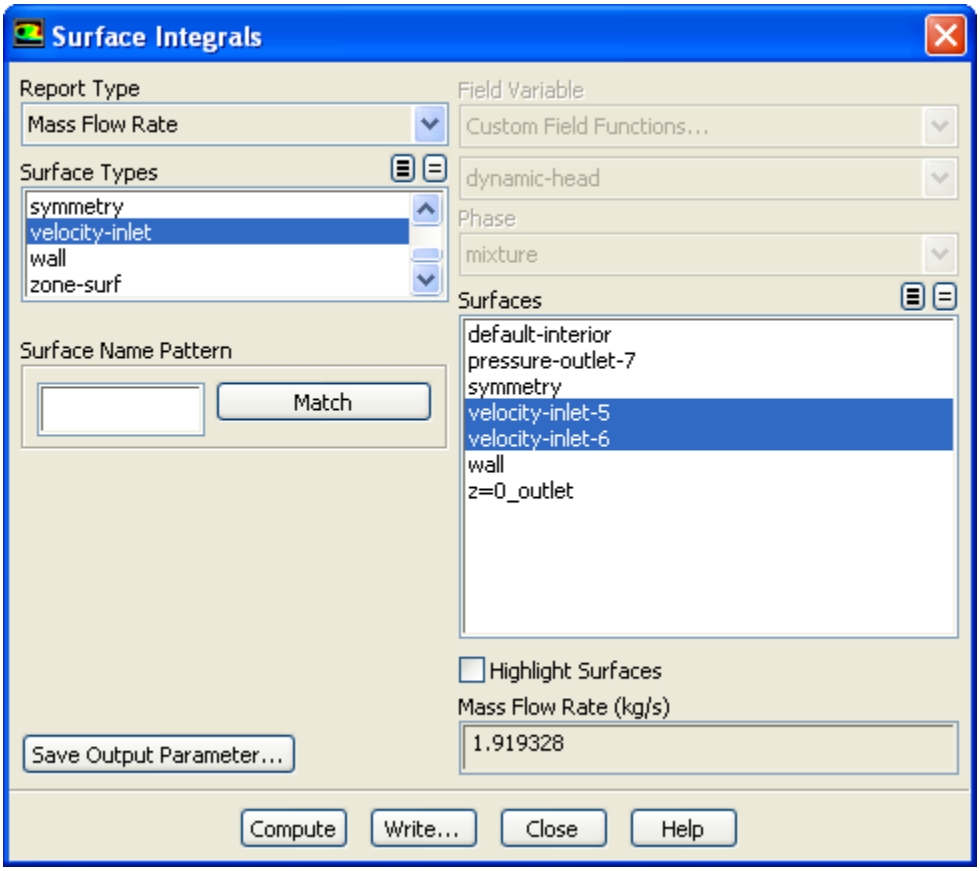
32.6.1. Generating a Surface Integral Report

To obtain a report for selected surfaces of the area or mass flow rate or the integral, flow rate, sum, facet maximum, facet minimum, vertex maximum, vertex minimum, or mass-, area-, facet-, or vertex-averaged quantity of a specified field variable, use the *Surface Integrals Dialog Box* (p. 2031) (*Figure 32.9* (p. 1520)).

Reports → **Surface Integrals** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.9 The Surface Integrals Dialog Box



The steps for generating the report are as follows:

1. Specify which type of report you are interested in by selecting **Area**, **Integral**, **Area-Weighted Average**, **Flow Rate**, **Mass Flow Rate**, **Mass-Weighted Average**, **Sum**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Vertex Average**, **Vertex Minimum**, or **Vertex Maximum** in the **Report Type** drop-down list.
2. If you are generating a report of area or mass flow rate, skip to the next step. Otherwise, use the **Field Variable** drop-down lists to select the field variable to be used in the surface integrations. First, select the desired category in the upper drop-down list. You can then select a related quantity from the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
3. In the **Surfaces** list, choose the surface(s) on which to perform the surface integration.

   If you want to select several surfaces of the same type, you can select that type in the **Surface Types** list instead. All of the surfaces of that type will be selected automatically in the **Surfaces** list (or deselected, if they are all selected already).

   Another shortcut is to specify a **Surface Name Pattern** and click **Match** to select surfaces with names that match the specified pattern. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (e.g., **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).
4. Click **Save Output Parameter...**. The *Save Output Parameter Dialog Box* (p. 2044) (*Figure 32.2* (p. 1512)) will open where you will specify the name of the newly created output parameter, or overwrite an existing

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

output parameter of the same type. The default report name format is report-type-n (e.g. surface-integral-1).

After the output parameter is created, it is listed in the *Parameters Dialog Box* (p. 2041). You can create any number of output parameters of this report type.

5. Click on the **Compute** button. Depending on the type of report you have selected, the label for the result will change to **Area**, **Integral**, **Area-Weighted Average**, **Flow Rate**, **Mass Flow Rate**, **Mass-Weighted Average**, **Sum of Facet Values**, **Average of Facet Values**, **Minimum of Facet Values**, **Maximum of Facet Values**, **Average of Surface Vertex Values**, **Minimum of Vertex Values**, or **Maximum of Vertex Values**, as appropriate. The computed results will also be printed in the ANSYS FLUENT console window.
6. To save the computed results to a file, click the **Write...** button and specify the filename in the resulting **Select File** dialog box.

Note the following items:

- Mass averaging "weights" toward regions of higher velocity (i.e., regions where more mass crosses the surface).
- Flow rates reported using the *Surface Integrals Dialog Box* (p. 2031) are not as accurate as those reported with the *Flux Reports Dialog Box* (p. 2027) (described in *Fluxes Through Boundaries* (p. 1510)).
- The facet and vertex average options are recommended for zero-area surfaces.

32.7. Volume Integration

For additional information about volume integrals, see Computing Volume Integrals in the Theory Guide.

For additional information, please see the following section:
32.7.1. Generating a Volume Integral Report

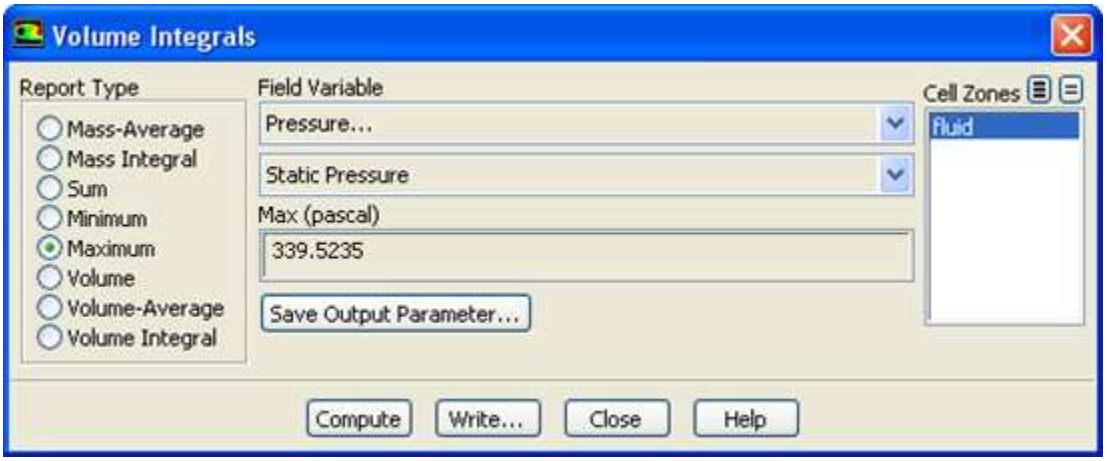
32.7.1. Generating a Volume Integral Report

To obtain a report (of quantities such as the volume, sum, minimum, maximum, volume integral, volume-weighted average, mass-weighted integral, or mass-weighted average) for selected cell zones for a specified field variable, use the *Volume Integrals Dialog Box* (p. 2034) (*Figure 32.10* (p. 1522)).

Reports → **Volume Integrals** → **Set Up...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.10 The Volume Integrals Dialog Box



The steps for generating the report are as follows:

1. Specify which type of report you are interested in by selecting **Volume**, **Sum**, **Max**, **Min**, **Volume Integral**, **Volume-Average**, **Mass Integral**, or **Mass-Average** under **Options**.
2. If you are generating a report of volume, skip to the next step. Otherwise, use the **Field Variable** drop-down lists to select the field variable to be used in the integral, sum, or averaged volume integrations. First, select the desired category in the upper drop-down list. You can then select a related quantity from the lower list. (See *Field Function Definitions* (p. 1529) for an explanation of the variables in the list.)
3. In the **Cell Zones** list, choose the zones on which to compute the volume, sum, max, min, volume integral, volume-weighted average, mass integral, or mass-averaged quantity.
4. Click **Save Output Parameter...**. The *Save Output Parameter Dialog Box* (p. 2044) (*Figure 32.2* (p. 1512)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. The default report name format is report-type-n (e.g. volume-integral-1).

   After the output parameter is created, it is listed in the *Parameters Dialog Box* (p. 2041). You can create any number of output parameters of this report type.
5. Click on the **Compute** button. Depending on the type of report you have selected, the label for the result will change to **Total Volume**, **Sum**, **Max**, **Min**, **Total Volume Integral**, **Volume-Weighted Average**, **Total Mass-Weighted Integral**, or **Mass-Weighted Average**, as appropriate.

   The computed results will also be printed in the ANSYS FLUENT console window.
6. To save the computed results to a file, click the **Write...** button and specify the filename in the resulting **Select File** dialog box.

32.8. Histogram Reports

In ANSYS FLUENT, you can print geometric and solution data in the console (text) window in histogram format or plot a histogram in the graphics window. Graphical display of histograms and the procedures for defining a histogram are discussed in *Histograms* (p. 1478).

The number of cells, the range of the selected variable or function, and the percentage of the total number of cells in the interval will be reported, as in the example below:

```
0 cells below 1.195482 (0 %)
2 cells between 1.195482 and 1.196048 (4.1666667 %)
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1 cells between 1.196048 and 1.196614 (2.0833333 %)
0 cells between 1.196614 and 1.19718 (0 %)
0 cells between 1.19718 and 1.197746 (0 %)
2 cells between 1.197746 and 1.198312 (4.1666667 %)
1 cells between 1.198312 and 1.198878 (2.0833333 %)
6 cells between 1.198878 and 1.199444 (12.5 %)
9 cells between 1.199444 and 1.20001 (18.75 %)
25 cells between 1.20001 and 1.200576 (52.083333 %)
2 cells between 1.200576 and 1.201142 (4.1666667
0 cells above 1.201142 (0 %)
```

To generate such a printed histogram, use the *Histogram Dialog Box* (p. 2015).

Reports → **Histogram** → **Set Up...**

Follow the instructions in *Histograms* (p. 1478) for generating histogram plots, but click on **Print** instead of **Plot** to create the report.

32.9. Discrete Phase

ANSYS FLUENT allows you to write particle states (position, velocity, diameter, temperature, and mass flow rate) to files at various boundaries and planes (lines in 2D) using the *Sample Trajectories Dialog Box* (p. 2036) (*Figure 25.30* (p. 1083)). Information about discrete phase reporting is discussed in detail in *Sampling of Trajectories* (p. 1082), *Histogram Reporting of Samples* (p. 1084), and *Summary Reporting of Current Particles* (p. 1085).

32.10. S2S Information

ANSYS FLUENT allows you to view the values of the view factor and radiation emitted from one zone to another. You will use the *S2S Information Dialog Box* (p. 2159) (*Figure 14.25* (p. 740)) to generate a report of these values. For details on reporting S2S information, refer to *Reporting Radiation in the S2S Model* (p. 739).

32.11. Reference Values

You can control the reference values that are used in the computation of derived physical quantities and non-dimensional coefficients. These reference values are used only for postprocessing.

Some examples of the use of reference values include the following:

- Force coefficients use the reference area, density, and velocity. In addition, the pressure force calculation uses the reference pressure.
- Moment coefficients use the reference length, area, density and velocity. In addition, the pressure force calculation uses the reference pressure.
- Reynolds number uses the reference length, density, and viscosity.
- Pressure and total pressure coefficients use the reference pressure, density, and velocity.
- Entropy uses the reference density, pressure, and temperature.
- Skin friction coefficient uses the reference density and velocity.
- Heat transfer coefficient uses the reference temperature.
- Turbomachinery efficiency calculations use the ratio of specific heats.

For additional information, please see the following sections:

32.11.1. Setting Reference Values
32.11.2. Setting the Reference Zone

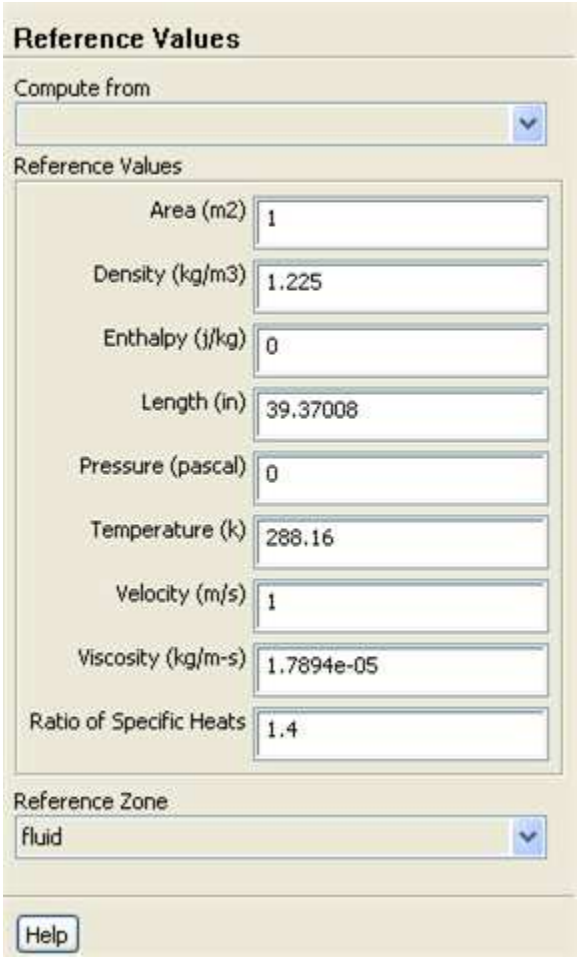
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.11.1. Setting Reference Values

To set the reference quantities used for computing normalized flow-field variables, use the *Reference Values Task Page* (p. 1898) (*Figure 32.11* (p. 1524)).

Reference Values

Figure 32.11 The Reference Values Task Page



You can input the reference values manually or compute them based on values of physical quantities at a selected boundary zone. The reference values to be set are **Area**, **Density**, **Enthalpy**, **Length**, **Pressure**, **Temperature**, **Velocity**, dynamic **Viscosity**, and **Ratio Of Specific Heats**.

For 2D problems, an additional quantity, **Depth**, can also be defined. This quantity will be used for reporting fluxes and forces, as well as relevant variables computed using the *Surface Integrals Dialog Box* (p. 2031) and the *Volume Integrals Dialog Box* (p. 2034) (e.g. **Area**, **Flow Rate**, **Mass Flow Rate**, **Volume**, etc.). You should verify that the value and units of **Depth** corresponds to the depth dimension of your application prior to reporting any of the variables above.

Important

The units for **Depth** are set independently from the units for **Length** in the *Set Units Dialog Box* (p. 1637).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to compute reference values from the conditions set on a particular boundary zone, select the zone in the **Compute From** drop-down list. Note, however, that depending on the boundary condition used, only some of the reference values may be set. For example, the reference length and area will not be set by computing the reference values from a boundary condition; you will need to set these manually.

To set the values manually, simply enter the value for each under the **Reference Values** heading.

32.11.2. Setting the Reference Zone

If you are solving a flow involving multiple reference frames or sliding meshes, you can plot velocities and other related quantities relative to the motion of a specified "reference zone". Choose the desired zone in the **Reference Zone** drop-down list. Changing the reference zone allows you to plot velocities (and total pressure, temperature, etc.) relative to the motion of different zones. See *Modeling Flows with Moving Reference Frames* (p. 513) for details about postprocessing of relative quantities.

32.12. Summary Reports of Case Settings

You may sometimes find it useful to get a report of the current settings in your case. In ANSYS FLUENT, you can list the settings for physical models, boundary conditions, material properties, and solver controls. This report allows you to get an overview of your current problem definition quickly, instead of having to check the settings in each dialog box.

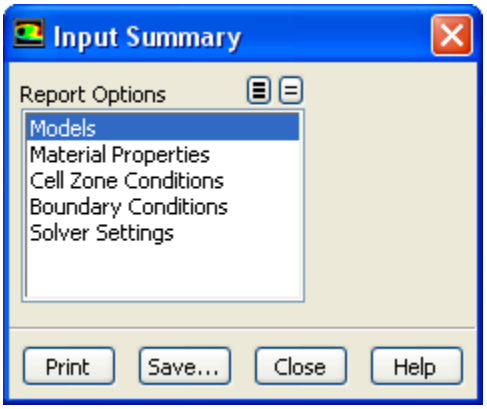
For additional information, please see the following section:
32.12.1. Generating a Summary Report

32.12.1. Generating a Summary Report

To generate a summary report you will use the *Input Summary Dialog Box* (p. 2158) (*Figure 32.12* (p. 1525)).

Report → **Input Summary...**

Figure 32.12 The Input Summary Dialog Box



The steps are as follows:

1. Select the information you would like to see in the report (**Models**, **Boundary Conditions**, **Solver Controls**, and/or **Material Properties**) in the **Report Options** list.
2. To print the information to the ANSYS FLUENT console window, click on the **Print** button. To save the information to a text file, click on the **Save...** button and specify the filename in the resulting **Select File** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.13. Memory and CPU Usage

There are two types of system reporting, which are accessed using the text interface, that can be performed while running ANSYS FLUENT processes:

- Reporting the status of each of the ANSYS FLUENT processes, including memory and CPU usage (`report/system/proc-stats`).
- Reporting the status of the machines where ANSYS FLUENT processes have been spawned, including memory and CPU status (`report/system/sys-stats`).

Important

Note that the `report/system/sys-stats` text command is only applicable for Windows (ntx86 and win64) and Linux (lnamd64, lns86, and lnia64) platforms. Similarly, the `report/system/proc-stats` text command is only applicable for Windows, Linux, and AIX platforms.

The type of information you can expect to see printed to the console when running in parallel, using the `report/system/proc-stats` text command, is

```
-----------------------------------------------------------------------------
       |  Mem  Usage  (MB)                |  CPU  Time  Usage  (Seconds)
ID     |  Current   Peak      Page  Fault  |  User      Kernel     Elapsed
-----------------------------------------------------------------------------
host   |  31.2422   285.242   9.439e+004   |  34.4531   1.90625    269.593
n0     |  525.949   743.438   3.933e+005   |  20.7656   3.70313    264.406
n1     |  516.063   737.438   3.867e+005   |  84.2813   166.328    264.437
-----------------------------------------------------------------------------
Total  |  1073.25   1766.12   8.744e+005   |  139.5     171.938    -
-----------------------------------------------------------------------------
```

Under `Mem Usage (MB)`

Current
is the virtual memory usage at this very moment.

Peak
is the peak virtual memory usage.

Page Fault
is the number of page faults that have occurred.

Under `CPU Time Usage (Seconds)`:

User
is the CPU time used by user processes.

Kernel
is the CPU time used by system kernel.

Elapsed
is the wall clock time elapsed since the process startup.

When using the `report/system/sys-stats` text command, where ANSYS FLUENT processes have been spawned on five machines, the following results are displayed:

```
-----------------------------------------------------------------------------
           |  CPU                          |  System  Mem  (MB)
Hostname   |  Number   Clock (MHz)  Load    |  Total      Available
-----------------------------------------------------------------------------
deva01     |  2        2211.38      0.2     |  32205.2    31479
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
deva03     |  2        2211.34      0       |  32205.2    21560.4
deva04     |  2        2211.34      0       |  16093.7    12075.8
deva05     |  2        2211.34      0       |  16093.7    14624.5
deva06     |  2        2211.34      0.07    |  16093.7    12095.4
-------------------------------------------------------------------------------
Total      |  10       -            -       |  112691     91835.2
-------------------------------------------------------------------------------
```

Under CPU

Number
is the number of processors on the machine.

Clock
is the processor speed.

Load
is the work load on the machine.

Under `System Mem (MB)`

Total
is the total system memory on the machine.

Available
is the available system memory on the machine.

You can use the two commands together to plan ANSYS FLUENT jobs and machines accordingly. It may also be useful to diagnose performance problems.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 33: Field Function Definitions

You must select flow variables for a number of tasks in ANSYS FLUENT. The values are computed and placed in temporary memory that is allocated for storing the results for each cell. For example, the **Compute** command associated with a dialog box that contains the field variable drop-down list calculates the values of the selected function and places them into temporary storage.

Node, Cell, and Facet Values (p. 1529) and *Velocity Reporting Options* (p. 1531) provide some general information related to the field variables. In *Field Variables Listed by Category* (p. 1532), the variables are listed by category in *Table 33.1:**Pressure** and **Density** Categories* (p. 1534) – *Table 33.14:**Acoustics** Category* (p. 1546). These tables will also indicate when each variable will be available. *Alphabetical Listing of Field Variables and Their Definitions* (p. 1547) contains an alphabetical listing of the variables along with their definitions. All variables appear as they would in the variable selection drop-down lists that are contained in many of the ANSYS FLUENT dialog boxes. *Custom Field Functions* (p. 1579) explains how you can calculate your own field function.

33.1. Node, Cell, and Facet Values
33.2. Velocity Reporting Options
33.3. Field Variables Listed by Category
33.4. Alphabetical Listing of Field Variables and Their Definitions
33.5. Custom Field Functions

33.1. Node, Cell, and Facet Values

For the following discussion, "surface" refers to a collection of facets, lines or points that are created and manipulated in the **Surface** menu. In most cases, these surfaces are created by computing intersections of constant isovalues with the domain cells or with existing surfaces.

For additional information, please see the following sections:

33.1.1. Cell Values
33.1.2. Node Values
33.1.3. Facet Values

33.1.1. Cell Values

ANSYS FLUENT stores most variables in cells. For postprocessing, the entire region contained within the cell has this value. A surface cell value is the value of the cell that has been intersected by a surface facet or line, or that contains a surface point. Since surface facets and lines are created from the intersection of isovalues and the existing mesh cells, this is a unique definition. Typically, the cell value on a boundary is the value in the cell adjacent to the boundary. For face-only functions like **Wall Shear Stress**, the cell value is the area-weighted average from the face values that define that cell as `c0`. This value is used for the cell values of postprocessing surfaces. But for boundary faces, the cell value actually displays/uses the exact face value.

33.1.2. Node Values

Node values are explicitly defined or obtained by weighted averaging of the cell data. Various boundary conditions impose values of field variables at the domain boundaries, so mesh node values on these

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

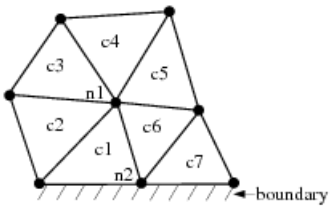
boundary zones are obtained by simple averaging of the adjacent boundary face data. In addition, for several variables (e.g., node coordinates) explicit node values are available at all nodes.

Computation of node values is performed in two steps:

1. Values at all nodes are initialized to the weighted average of the surrounding cell values. The weights are the inverses of the distances between the nodes and the cell centroid.
2. At boundaries, these node values are overwritten with the simple average of the boundary face values. Variables for which explicit node values are available at boundaries are indicated by *bnv* in *Table 33.1:**Pressure** and **Density** Categories* (p. 1534) – *Table 33.14:**Acoustics** Category* (p. 1546).

For example, in *Figure 33.1* (p. 1530), the value at node $n1$ will be computed from the weighted average of the values in the surrounding cells ($c1$– $c6$). The value at node $n2$ will be computed from the simple average of the boundary faces ($bf1$ and $bf2$) if there are explicit boundary values available for the variable in question.

Figure 33.1 Computing Node Values



Important

Note that explicit boundary node values are not available for custom field functions.

33.1.2.1. Vertex Values for Points That are Not Mesh Nodes

The values of the nodes on surfaces are linearly interpolated from the mesh node data. For zone surfaces, the nodes on the surface and the zone correspond; thus, the values are identical. For surfaces that are not zone surfaces (e.g., isosurfaces, plane surfaces, etc.), the node values are interpolated from mesh nodes on the cell faces intersected by the postprocessing surface. For point surfaces, the value is interpolated from all the mesh nodes of the cell containing the point.

33.1.3. Facet Values

Facets can be created on preprocessing surfaces and postprocessing surfaces.

33.1.3.1. Facet Values on Zone Surfaces

The interior facets on a zone surface are associated with two cells (`c0` and `c1`). The values of a specified variable on such facets are computed as the average of the two cell values of the selected variable.

The boundary facet values of a specified field variable on a zone surface are computed from the boundary condition provided by the user.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.1.3.2. Facet Values on Postprocessing Surfaces

Each facet on a postprocessing surface is associated with a cell. The values of a specified variable on facets are the same as the cell values of the selected variable in the associated cells (this includes iso surfaces, planes, lines, points, rakes, quadric, etc.).

33.2. Velocity Reporting Options

The following methods are available for reporting velocities:

- Cartesian velocities:

  These velocities are based on the Cartesian coordinate system used by the geometry. To report Cartesian velocities, select **X Velocity**, **Y Velocity**, or **Z Velocity**. This is the most common type of velocity reported.

- Cylindrical velocities:

  These velocities are the axial, radial, and tangential components based on the following coordinate systems:

  – For axisymmetric problems, in which the rotation axis must be the x axis, the x direction is the axial direction and the y direction is the radial direction. (If you model axisymmetric swirl, the swirl direction is the tangential direction.)

  – For 2D problems involving a single cell zone, the z direction is the axial direction, and its origin is specified in the *Fluid Dialog Box* (p. 1796).

  – For 3D problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the *Fluid Dialog Box* (p. 1796).

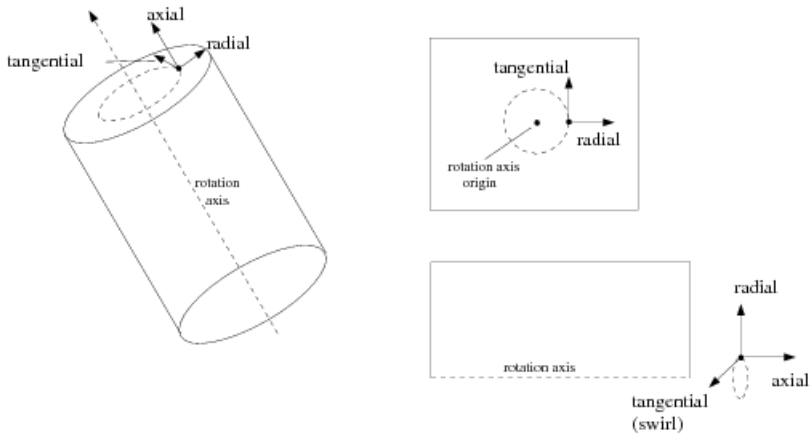
  – For problems involving multiple zones (e.g., multiple reference frames or sliding meshes), the coordinate system is defined by the rotation axis specified in the *Fluid Dialog Box* (p. 1796) (or *Solid Dialog Box* (p. 1802)) for the "reference zone". The reference zone is chosen in the *Reference Values Task Page* (p. 1898), as described in *Reference Values* (p. 1523). Recall that for 2D problems, you will specify only the axis origin; the z direction is always the axial direction.

  For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially out from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis.

  To report cylindrical velocities, select **Axial Velocity**, **Radial Velocity**, etc. *Figure 33.2* (p. 1532) illustrates the cylindrical velocities available for different types of domains. For 3D problems, you can report axial, radial, and tangential velocities. For 2D problems, radial and tangential velocities are available. For axisymmetric problems, you can report axial and radial velocities, and, if you are modeling axisymmetric swirl, you can also report the swirl velocity (which is equivalent to the tangential velocity).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.2 Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains



- Relative velocities: These velocities are based on the coordinate system and motion of a moving reference frame. They are useful when you are modeling your flow using a moving reference frame, a mixing plane, multiple reference frames, or sliding meshes. (See *Modeling Flows with Moving Reference Frames* (p. 513) for information about modeling flow in moving zones.) To report relative velocities, select **Relative X Velocity**, **Relative Y Velocity**, **Relative Radial Velocity**, etc. (Note that you can report relative velocities for both Cartesian and cylindrical components.)

  If you are using a single moving reference frame, the relative velocity values will be reported with respect to the moving frame. If you are using multiple reference frames, mixing planes, or sliding meshes, you will need to specify the frame to which you want the velocities to be relative by choosing the appropriate cell zone as the **Reference Zone** in the *Reference Values Task Page* (p. 1898) (see *Reference Values* (p. 1523)). The axis of rotation for each cell zone is defined in the associated *Fluid Dialog Box* (p. 1796) or *Solid Dialog Box* (p. 1802). (See *Specifying the Rotation Axis* (p. 217) or *Specifying the Rotation Axis* (p. 219) for details.)

  Note that if your problem does not involve any moving zones, relative and absolute velocities will be equivalent.

Note that relative velocities can also be used to compute stagnation quantities (total pressure and total temperature), and that the cylindrical coordinate systems described in the second item above are used for defining the **Axial Coordinate** and **Radial Coordinate** as well.

33.3. Field Variables Listed by Category

In *Table 33.1:**Pressure** and **Density** Categories* (p. 1534) – *Table 33.14:**Acoustics** Category* (p. 1546), the following restrictions apply to marked variables:

2d	available only for 2D flows
2da	available only for 2D axisymmetric flows (with or without swirl)
2dasw	available only for 2D axisymmetric swirl flows
3d	available only for 3D flows
ark	available only with the Aungier-Redlich-Kwong real gas model
bns	available only for broadband noise source models
bnv	node values available at boundaries
cpl	available only in the density-based solvers
cv	available only for cell values (**Node Values** option turned off)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

des	available only when the DES turbulence model is used
dil	not available with full multicomponent diffusion
do	available only when the discrete ordinates radiation model is used
dpm	available only for coupled discrete phase calculations
dtrm	available only when the discrete transfer radiation model is used
fwh	available only with the Ffowcs Williams and Hawkings acoustics model
e	available only for energy calculations
edc	available only with the EDC model for turbulence-chemistry interaction
emm	available also when the Eulerian multiphase model is used
ewt	available only with the enhanced wall treatment
fv	available for face values
gran	available only if a granular phase is present
h2o	available only when the mixture contains water
id	available only when the ideal gas law is enabled for density
ke	available only when one of the k- ε turbulence models is used
kklo	available only when the k- kl- ω model is used
kw	available only when one of the k- ω turbulence models is used
les	available only when the LES turbulence model is used
melt	available only when the melting and solidification model is used
mix	available only when the multiphase mixture model is used
mp	available only for multiphase models
nox	available only for NOx calculations
np	not available in parallel solvers
nv	uses explicit node value function
p	available only in parallel solvers
p1	available only when the P-1 radiation model is used
pdf	available only for non-premixed combustion calculations
pmx	available only for premixed combustion calculations
ppmx	available only for partially premixed combustion calculations
r	available only when the Rosseland radiation model is used
rad	available only for radiation heat transfer calculations
rc	available only for finite-rate reactions
rg	available only for the real gas models
rsm	available only when the Reynolds stress turbulence model is used
s2s	available only when the surface-to-surface radiation model is used
sa	available only when the Spalart-Allmaras turbulence model is used
seg	available only in the pressure-based solver
sp	available only for species calculations
sr	available only for surface reactions
sol	available only when the solar model is used

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

soot	available only for soot calculations
sst	available only when the SST Transition model is used
stat	available only with data sampling for transient statistics
stcm	available only for stiff chemistry calculations
t	available only for turbulent flows
turbo	available only when a turbomachinery topology has been defined
udm	available only when a user-defined memory is used
uds	available only when a user-defined scalar is used
v	available only for viscous flows

Table 33.1 Pressure and Density Categories

Category	Variable
Pressure...	**Static Pressure** (bnv)
	Pressure Coefficient
	Dynamic Pressure
	Absolute Pressure (bnv)
	Total Pressure (bnv)
	Relative Total Pressure
Density...	**Density**
	Density All

Table 33.2 Velocity Category

Category	Variable
Velocity...	**Velocity Magnitude** (bnv)
	X Velocity (bnv)
	Y Velocity (bnv)
	Z Velocity (3d, bnv)
	Swirl Velocity (2dasw, bnv)
	Axial Velocity (2da or 3d)
	Radial Velocity
	Stream Function (2d)
	Tangential Velocity
	Mach Number (id or rg)
	Relative Velocity Magnitude (bnv)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Relative X Velocity (bnv)
	Relative Y Velocity (bnv)
	Relative Z Velocity (3d, bnv)
	Relative Axial Velocity (2da)
	Relative Radial Velocity (2da)
	Relative Swirl Velocity (2dasw, bnv)
	Relative Tangential Velocity
	Relative Mach Number (id or rg)
	Mesh X-Velocity (nv)
	Mesh Y-Velocity (nv)
	Mesh Z-Velocity (3d, nv)
	Velocity Angle
	Relative Velocity Angle
	Vorticity Magnitude (v)
	Helicity (v, 3d)
	X-Vorticity (v, 3d)
	Y-Vorticity (v, 3d)
	Z-Vorticity (v, 3d)
	Cell Reynolds Number (v)
	Preconditioning Reference Velocity (cpl)

Table 33.3 Temperature, Radiation, and Solidification/Melting Categories

Category	Variable
Temperature...	**Static Temperature** (e, bnv, nv)
	Total Temperature (e, nv)
	Sensible Enthalpy (e, nv)
	Enthalpy (e, nv)
	Relative Total Temperature (e)
	Rothalpy (e, nv)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Fine Scale Temperature (edc,e) **Wall Temperature (Outer Surface)** (fv, e, v) **Wall Temperature (Inner Surface)** (fv, e, v) **Total Enthalpy** (e) **Total Enthalpy Deviation** (e) **Entropy** (e) **Total Energy** (e) **Internal Energy** (e)
Radiation...	**Absorption Coefficient** (r, p1, do, or dtrm) **Scattering Coefficient** (r, p1, or do) **Refractive Index** (do) **Radiation Temperature** (p1 or do) **Incident Radiation** (p1 or do) **Incident Radiation (Band n)** (p1 (non-gray) or do (non-gray)) **Surface Cluster ID** (fv, s2s)
Solidification/ Melting	**Liquid Fraction** (melt) **Contact Resistivity** (fv, melt) **X Pull Velocity** (melt (if calculated)) **Y Pull Velocity** (melt (if calculated)) **Z Pull Velocity** (melt (if calculated), 3d) **Axial Pull Velocity** (melt (if calculated), 2da) **Radial Pull Velocity** (melt (if calculated), 2da) **Swirl Pull Velocity** (melt (if calculated), 2dasw)

Table 33.4 Turbulence Category

Category	Variable
Turbulence...	**Turbulent Kinetic Energy (k)** (ke, kw, or rsm; bnv, nv, or emm) **Laminar Kinetic Energy** (kklo) **Turbulent Intensity** (ke, kw, or rsm) **Intermittency** (sst) **Momentum Thickness Re** (sst)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	UU Reynolds Stress (rsm; emm)
	VV Reynolds Stress (rsm; emm)
	WW Reynolds Stress (rsm; emm)
	UV Reynolds Stress (rsm; emm)
	UW Reynolds Stress (rsm, 3d; emm)
	VW Reynolds Stress (rsm, 3d; emm)
	Turbulent Dissipation Rate (Epsilon) (ke or rsm; bnv, nv, or emm)
	Specific Dissipation Rate (Omega) (kw)
	Production of k (ke, kw, or rsm; emm)
	Modified Turbulent Viscosity (sa)
	Turbulent Viscosity (sa, ke, kw, rsm, or des)
	Effective Viscosity (sa, ke, kw, rsm, or des; emm)
	Turbulent Viscosity Ratio (ke, kw, rsm, sa, or des; emm)
	Subgrid Kinetic Energy (les)
	Subgrid Turbulent Viscosity (les)
	Subgrid Effective Viscosity (les)
	Subgrid Turbulent Viscosity Ratio (les)
	Subgrid Filter Length (les)
	Effective Thermal Conductivity (t, e)
	Effective Prandtl Number (t, e)
	Wall Ystar (fv, ke, kw, or rsm)
	Wall Yplus (fv, t)
	Turbulent Reynolds Number (Re_y) (ke or rsm; ewt)
	Relative Length Scale (DES) (des)

Table 33.5 Species, Reactions, Pdf, and Premixed Combustion Categories

Category	Variable
Species...	**Mass fraction of species-n** (sp, pdf, or ppmx; nv)
	Mole fraction of species-n (sp, pdf, or ppmx)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Molar Concentration of species-n (sp, pdf, or ppmx) **Lam Diff Coef of species-n** (sp, dil) **Eff Diff Coef of species-n** (t, sp, dil) **Thermal Diff Coef of species-n** (sp) **Enthalpy of species-n** (sp) **species-n Source Term** (rc, cpl) **Surface Deposition Rate of species-n** (sr) **Surface Coverage of species-n** (sr) **Relative Humidity** (sp, pdf, or ppmx; h2o) **Time Step Scale** (sp, stcm) **Fine Scale Mass fraction of species-n** (edc) **Cell Time Scale** (edc) **EDC Cell Volume Fraction** (edc)
Reactions...	**Rate of Reaction-n** (rc) **Kinetic Rate of Reaction-n** (rc) **Turbulent Rate of Reaction-n** (rc, t) **Heat of Reaction** (e, rc) **Net Reaction Rate of Species-n** (edc, stcm)
Pdf...	**Mean Mixture Fraction** (pdf or ppmx; nv) **Secondary Mean Mixture Fraction** (pdf or ppmx; nv) **Mixture Fraction Variance** (pdf or ppmx; nv) **Secondary Mixture Fraction Variance** (pdf or ppmx; nv) **Fvar Prod** (pdf or ppmx) **Fvar2 Prod** (pdf or ppmx) **Scalar Dissipation** (pdf or ppmx) **PDF Table Adiabatic Enthalpy** (pdf or ppmx) **PDF Table Heat Loss/Gain** (e, pdf or ppmx)
Premixed Combustion...	**Progress Variable** (pmx or ppmx; nv) **Damkohler Number** (pmx or ppmx)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Stretch Factor (pmx or ppmx)
	Turbulent Flame Speed (pmx or ppmx)
	Static Temperature (pmx or ppmx)
	Product Formation Rate (pmx or ppmx)
	Laminar Flame Speed (pmx or ppmx)
	Critical Strain Rate (pmx or ppmx)
	Adiabatic Flame Temperature (pmx or ppmx)
	Unburnt Fuel Mass Fraction (pmx or

Table 33.6 NOx, Soot, and Unsteady Statistics Categories

Category	Variable
NOx...	**Mass fraction of NO** (nox)
	Mass fraction of HCN (nox)
	Mass fraction of NH3 (nox)
	Mass fraction of N2O (nox)
	Mole fraction of NO (nox)
	Mole fraction of HCN (nox)
	Mole fraction of NH3 (nox)
	Mole fraction of N2O (nox)
	NO Density (nox)
	HCN Density (nox)
	NH3 Density (nox)
	N2O Density (nox)
	Variance of Temperature (nox)
	Variance of Species (nox)
	Variance of Species 1 (nox)
	Variance of Species 2 (nox)
	Rate of NO (nox)
	Rate of Thermal NO (nox)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Rate of Prompt NO (nox) **Rate of Fuel NO** (nox) **Rate of N2OPath NO** (nox) **Rate of Reburn NO** (nox) **Rate of SNCR NO** (nox) **Rate of USER NO** (nox)
Soot...	**Mass fraction of soot** (soot) **Mass fraction of Nuclei** (soot) **Mole fraction of soot** (soot) **Soot Density** (soot) **Rate of Soot** (soot) **Rate of Nuclei** (soot)
Unsteady Statistics...	**Mean quantity-n** (stat) **RMS quantity-n** (stat)

Table 33.7 Phases, Discrete Phase Model, Granular Pressure, and Granular Temperature Categories

Category	Variable
Phase Interaction...	**Heterogeneous Reaction Rate n**
Phases...	**Volume fraction** (mp)
Discrete Phase Model...	**DPM Mass Source** (dpm) **DPM Erosion** (dpm, cv, fv) **DPM Accretion** (dpm, cv, fv) **DPM X Momentum Source** (dpm) **DPM Y Momentum Source** (dpm) **DPM Z Momentum Source** (dpm, 3d) **DPM Swirl Momentum Source** (dpm, 2dasw) **DPM Sensible Enthalpy Source** (dpm, e, rc) **DPM Enthalpy Source** (dpm, e) **DPM Absorption Coefficient** (dpm, rad) **DPM Emission** (dpm, rad)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	DPM Scattering (dpm, rad) **DPM Burnout** (dpm, sp, e) **DPM Evaporation/Devolatilization** (dpm, sp, e) **DPM Concentration** (dpm) **DPM species-n Source** (dpm, sp, e)
Granular Pressure...	**Granular Pressure** (emm, gran)
Granular Temperature...	**Granular Temperature** (emm, gran)

Table 33.8 Properties Category

Category	Variable
Properties...	**Molecular Viscosity** (v) **Diameter** (mix, emm) **Granular Conductivity** (mix, emm, gran) **Thermal Conductivity** (e, v) **Specific Heat (Cp)** (e) **Specific Heat Ratio (gamma)** (id) **Gas Constant (R)** (id or rg) **Molecular Prandtl Number** (e, v) **Mean Molecular Weight** (seg, pdf) **Sound Speed** (id or rg) **Compressibility Factor** (rg) **Reduced Temperature** (ark) **Reduced Pressure** (ark) **Critical Temperature** (ark, spe) **Critical Pressure** (ark, spe) **Acentric Factor** (ark,spe) **Critical Specific Volume** (ark,spe)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Spinodal Temperature (ark)

Table 33.9 Wall Fluxes, User Defined Scalars, and User Defined Memory Categories

Category	Variable
Wall Fluxes...	**Wall Shear Stress** (v, emm, fv)
	X-Wall Shear Stress (v, emm, fv)
	Y-Wall Shear Stress (v, emm, fv)
	Z-Wall Shear Stress (v, 3d, emm, fv)
	Axial-Wall Shear Stress (2da, fv)
	Radial-Wall Shear Stress (2da, fv)
	Swirl-Wall Shear Stress (2dasw, fv)
	Skin Friction Coefficient (v, emm, fv)
	Total Surface Heat Flux (e, v, fv)
	Radiation Heat Flux (fv, rad)
	Solar Heat Flux (sol, fv)
	Absorbed Radiation Flux (Band-n) (do, fv)
	Absorbed Visible Solar Flux (sol, fv)
	Absorbed IR Solar Flux (sol, fv)
	Reflected Radiation Flux (Band-n) (do, fv)
	Reflected Visible Solar Flux (sol, fv)
	Reflected IR Solar Flux (sol, fv)
	Transmitted Radiation Flux (Band-n) (do, fv)
	Transmitted Visible Solar Flux (sol, fv)
	Transmitted IR Solar Flux (sol, fv)
	Beam Irradiation Flux (Band-n) (do, fv)
	Surface Incident Radiation (do, dtrm, or s2s; fv)
	Surface Heat Transfer Coef. (e, v, fv)
	Wall Func. Heat Tran. Coef. (e, v, fv)
	Surface Nusselt Number (e, v, fv)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Surface Stanton Number (e, v, fv)
User Defined Scalars...	**Scalar-n** (uds) **Diffusion Coef. of Scalar-n** (uds)
User Defined Memory...	**udm-n** (udm)

Table 33.10 Cell Info and Mesh Categories

Category	Variable
Cell Info...	**Cell Partition** (np) **Active Cell Partition** (p) **Stored Cell Partition** (p) **Cell Id** (p) **Cell Element Type** **Cell Zone Type** **Cell Zone Index** **Partition Neighbors**
Mesh...	**X-Coordinate** (nv) **Y-Coordinate** (nv) **Z-Coordinate** (3d, nv) **Axial Coordinate** (nv) **Angular Coordinate** (3d, nv) **Abs. Angular Coordinate** (3d, nv) **Radial Coordinate** **Face Area Magnitude** **X Face Area** **Y Face Area** **Z Face Area** (3d) **Orthogonal Quality** **Cell Equiangle Skew** **Cell Equivolume Skew** **Cell Volume**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	2D Cell Volume (2da)
	Cell Wall Distance
	Face Handedness
	Cell Volume Derivative
	Cell Volume Error
	Dynamic Cell Volume

Table 33.11 Mesh Category (Turbomachinery-Specific Variables) and Adaption Category

Category	Variable
Mesh...	**Meridional Coordinate** (nv, turbo)
	Abs Meridional Coordinate (nv, turbo)
	Spanwise Coordinate (nv, turbo)
	Abs (H-C) Spanwise Coordinate (nv, turbo)
	Abs (C-H) Spanwise Coordinate (nv, turbo)
	Pitchwise Coordinate (nv, turbo)
	Abs Pitchwise Coordinate (nv, turbo)
Adaption...	**Adaption Function**
	Adaption Curvature
	Adaption Space Gradient
	Adaption Iso-Value
	Existing Value
	Boundary Cell Distance
	Boundary Normal Distance
	Boundary Volume Distance (np)
	Cell Volume Change
	Cell Surface Area
	Cell Warpage
	Cell Children

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Cell Refine

Table 33.12 Residuals Category

Category	Variable
Residuals...	**Mass Imbalance** (seg)
	Pressure Residual (cpl)
	X-Velocity Residual (cpl)
	Y-Velocity Residual (cpl)
	Z-Velocity Residual (cpl, 3d)
	Axial-Velocity Residual (cpl, 2da)
	Radial-Velocity Residual (cpl, 2da)
	Swirl-Velocity Residual (cpl, 2dasw)
	Temperature Residual (cpl, e)
	Species-n Residual (cpl, sp)
	Time Step (cpl)
	Pressure Correction (cpl)
	X-Velocity Correction (cpl)
	Y-Velocity Correction (cpl)
	Z-Velocity Correction (cpl, 3d)
	Axial-Velocity Correction (cpl, 2da)
	Radial-Velocity Correction (cpl, 2da)
	Swirl-Velocity Correction (cpl, 2dasw)
	Temperature Correction (cpl, e)
	Species-n Correction (cpl,

Table 33.13 Derivatives Category

Category	Variable
Derivatives...	**Strain Rate** (v)
	dX-Velocity/dx
	dY-Velocity/dx

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	dZ-Velocity/dx (3d)
	dAxial-Velocity/dx (2da)
	dRadial-Velocity/dx (2da)
	dSwirl-Velocity/dx (2dasw)
	d species-n/dx (cpl, sp)
	dX-Velocity/dy
	dY-Velocity/dy
	dZ-Velocity/dy (3d)
	dAxial-Velocity/dy (2da)
	dRadial-Velocity/dy (2da)
	dSwirl-Velocity/dy (2dasw)
	d species-n/dy (cpl, sp)
	dX-Velocity/dz (3d)
	dY-Velocity/dz (3d)
	dZ-Velocity/dz (3d)
	d species-n/dz (cpl, sp, 3d)
	dOmega/dx (2dasw)
	dOmega/dy (2dasw)
	dp-dX (seg)
	dp-dY (seg)
	dp-dZ (seg,

Table 33.14 Acoustics Category

Category	Variable
Acoustics...	**Surface dpdt RMS** (fv, fwh)
	Acoustic Power Level (dB) (bns)
	Acoustic Power (bns)
	Jet Acoustic Power Level (dB) (bns, 2da)
	Jet Acoustic Power (bns, 2da)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Surface Acoustic Power Level (dB) (bns, fv)
	Surface Acoustic Power (bns, fv)
	Lilley's Self-Noise Source (bns)
	Lilley's Shear-Noise Source (bns)
	Lilley's Total Noise Source (bns)
	LEE Self-Noise X-Source (bns)
	LEE Shear-Noise X-Source (bns)
	LEE Total Noise X-Source (bns)
	LEE Self-Noise Y-Source (bns)
	LEE Shear-Noise Y-Source (bns)
	LEE Total Noise Y-Source (bns)
	LEE Self-Noise Z-Source (bns, 3d)
	LEE Shear-Noise Z-Source (bns, 3d)
	LEE Total Noise Z-Source (bns, 3d)

33.4. Alphabetical Listing of Field Variables and Their Definitions

Below, the variables listed in *Table 33.1:**Pressure** and **Density** Categories* (p. 1534) – *Table 33.14:**Acoustics** Category* (p. 1546) are defined. For some variables (such as residuals) a general definition is given under the category name, and variables in the category are not listed individually. When appropriate, the unit quantity is included, as it appears in the **Quantities** list in the *Set Units Dialog Box* (p. 1637).

Abs (C-H) Spanwise Coordinate
(in the **Mesh...** category) is the dimensional coordinate in the spanwise direction, from casing to hub. Its unit quantity is **length**.

Abs (H-C) Spanwise Coordinate
(in the **Mesh...** category) is the dimensional coordinate in the spanwise direction, from hub to casing. Its unit quantity is **length**.

Abs Meridional Coordinate
(in the **Mesh...** category) is the dimensional coordinate that follows the flow path from inlet to outlet. Its unit quantity is **length**.

Abs Pitchwise Coordinate
(in the **Mesh...** category) is the dimensional coordinate in the circumferential (pitchwise) direction. Its unit quantity is **angle**.

Absolute Pressure
(in the **Pressure...**category) is equal to the operating pressure plus the gauge pressure. See *Operating Pressure* (p. 449) for details. Its unit quantity is **pressure**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absorbed Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux absorbed by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Absorbed Visible Solar Flux, Absorbed IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux absorbed by a semi-transparent wall or porous jump boundary for a visible or infrared (IR) radiation.

Absorption Coefficient

(in the **Radiation...** category) is the property of a medium that describes the amount of absorption of thermal radiation per unit path length within the medium. It can be interpreted as the inverse of the mean free path that a photon will travel before being absorbed (if the absorption coefficient does not vary along the path). The unit quantity for **Absorption Coefficient** is **length-inverse**.

Acentric Factor

(in the **Properties...** category) is the mixture acentric factor. This property is available when a composition dependent option is selected for acentric factor in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Acoustic Power

(in the **Acoustics...** category) is the acoustic power per unit volume generated by isotropic turbulence (see Equation 15–10 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used. Its unit quantity is **power** per **volume**.

Acoustic Power Level (dB)

(in the **Acoustics...** category) is the acoustic power per unit volume generated by isotropic turbulence and reported in dB (see Equation 15–13 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Active Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of ANSYS FLUENT, the partition ID can be used to determine the extent of the various groups of cells. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. See *Partitioning the Mesh Manually and Balancing the Load* (p. 1605) for more information.

Adaption...

includes field variables that are commonly used for adapting the mesh. For information about solution adaption, see *Adapting the Mesh* (p. 1325).

Adaption Function

(in the **Adaption...** category) can be either the **Adaption Space Gradient** or the **Adaption Curvature**, depending on the settings in the *Gradient Adaption Dialog Box* (p. 2119). For instance, the **Adaption Curvature** is the undivided Laplacian of the values in temporary cell storage. To display contours of the Laplacian of pressure, for example, you first select **Static Pressure**, click the **Compute** (or **Display**) button, select **Adaption Function**, and finally click the **Display** button.

Adaption Iso-Value

(in the **Adaption...** category) is the desired field variable function.

Adaption Space Gradient

(in the **Adaption...** category) is the first derivative of the desired field variable.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$|e_{i1}| = (A_{cell})^{\frac{r}{2}} |\nabla f| \quad \textbf{(33–1)}$$

Depending on the settings in the *Gradient Adaption Dialog Box* (p. 2119), this equation will either be scaled or normalized. Recommended for problems with shock waves (i.e., supersonic, inviscid flows). For more information, see Gradient Adaption Approach in the Theory Guide.

Adaption Curvature

(in the **Adaption...** category) is the second derivative of the desired field variable.

$$|e_{i2}| = (A_{cell})^{\frac{r}{2}} |\nabla^2 f| \quad \textbf{(33–2)}$$

Depending on the settings in the *Gradient Adaption Dialog Box* (p. 2119), this equation will either be scaled or normalized. Recommended for smooth solutions (i.e., viscous, incompressible flows). For more information, see Gradient Adaption Approach in the Theory Guide.

Adiabatic Flame Temperature

(in the **Premixed Combustion...** category) is the adiabatic temperature of burnt products in a laminar premixed flame (T_b in Equation 9–67 in the Theory Guide). Its unit quantity is **temperature**.

Angular Coordinate

(in the **Mesh...** category) is the angle between the radial vector and the position vector. The radial vector is obtained by transforming the default radial vector (y-axis) by the same rotation that was applied to the default axial vector (z-axis). This assumes that, after the transformation, the default axial vector (z-axis) becomes the reference axis. The angle is positive in the direction of cross-product between reference axis and radial vector.

Abs. Angular Coordinate

(in the **Mesh...** category) is the absolute value of the **Angular Coordinate** defined above.

Axial Coordinate

(in the **Mesh...** category) is the distance from the origin in the axial direction. The axis origin and (in 3D) direction is defined for each cell zone in the *Fluid Dialog Box* (p. 1796) or *Solid Dialog Box* (p. 1802). The axial direction for a 2D model is always the z direction, and the axial direction for a 2D axisymmetric model is always the x direction. The unit quantity for **Axial Coordinate** is **length**.

Axial Pull Velocity

(in the **Solidification/Melting...** category) is the axial-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Axial Velocity

(in the **Velocity...** category) is the component of velocity in the axial direction. (See *Velocity Reporting Options* (p. 1531) for details.) For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. Its unit quantity is **velocity**.

Axial-Wall Shear Stress

(in the **Wall Fluxes...** category) is the axial component of the force acting tangential to the surface due to friction. Its unit quantity is **pressure**.

Beam Irradiation Flux (Band-b)

(in the **Wall Fluxes...** category) is specified as an incident heat flux (W/m^2) for each wavelength band.

Boundary Cell Distance

(in the **Adaption...** category) is an integer that indicates the approximate number of cells from a boundary zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Normal Distance

(in the **Adaption...** category) is the distance of the cell centroid from the closest boundary zone.

Boundary Volume Distance

(in the **Adaption...** category) is the cell volume distribution based on the **Boundary Volume**, **Growth Factor**, and normal distance from the selected **Boundary Zones** defined in the *Boundary Adaption Dialog Box* (p. 2117). See *Boundary Adaption* (p. 1328) for details.

Cell Children

(in the **Adaption...** category) is a binary identifier based on whether a cell is the product of a cell subdivision in the hanging-node adaption process (value = 1) or not (value = 0).

Cell Element Type

(in the **Cell Info...** category) is the integer cell element type identification number. Each cell can have one of the following element types:

```
triangle      1
tetrahedron  2
quadrilateral 3
hexahedron   4
pyramid     5
wedge      6
```

Cell Equiangle Skew

(in the **Mesh...** category) is a nondimensional parameter calculated using the normalized angle deviation method, and is defined as

$$\max\left[\frac{q_{max}-q_e}{180-q_e}, \frac{q_e-q_{min}}{q_e}\right] \qquad \textbf{(33–3)}$$

where

q_{max} = largest angle in the face or cell

q_{min} = smallest angle in the face or cell

q_e = angle for an equiangular face or cell (e.g., 60 for a triangle and 90 for a square)

A value of 0 indicates a best case equiangular cell, and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D). **Cell Equiangle Skew** applies to all elements.

Cell Equivolume Skew

(in the **Mesh...** category) is a nondimensional parameter calculated using the volume deviation method, and is defined as

$$\frac{(\text{optimal-cell-size})-(\text{cell-size})}{(\text{optimal-cell-size})} \qquad \textbf{(33–4)}$$

where optimal-cell-size is the size of an equilateral cell with the same circumradius. A value of 0 indicates a best case equilateral cell and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D). **Cell Equivolume Skew** applies only to triangular and tetrahedral elements.

Cell Id

(in the **Cell Info...** category) is a unique integer identifier associated with each cell.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Info...

includes quantities that identify the cell and its relationship to other cells.

Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of ANSYS FLUENT, the partition ID can be used to determine the extent of the various groups of cells.

Cell Refine Level

(in the **Adaption...** category) is an integer that indicates the number of times a cell has been subdivided in the hanging node adaption process, compared with the original mesh. For example, if one quad cell is split into four quads, the **Cell Refine Level** for each of the four new quads will be 1. If the resulting four quads are split again, the **Cell Refine Level** for each of the resulting 16 quads will be 2.

Cell Reynolds Number

(in the **Velocity...** category) is the value of the Reynolds number in a cell. (Reynolds number is a dimensionless parameter that is the ratio of inertia forces to viscous forces.) **Cell Reynolds Number** is defined as

$$Re \equiv \frac{\rho u d}{\mu} \tag{33–5}$$

where ρ is density, u is velocity magnitude, μ is the effective viscosity (laminar plus turbulent), and d is **Cell Volume** $^{1/2}$ for 2D cases and **Cell Volume** $^{1/3}$ in 3D or axisymmetric cases.

Cell Surface Area

(in the **Adaption...** category) is the total surface area of the cell, and is computed by summing the area of the faces that compose the cell.

Cell Volume

(in the **Mesh...** category) is the volume of a cell. In 2D the volume is the area of the cell multiplied by the unit depth. For axisymmetric cases, the cell volume is calculated using a reference depth of 1 radian. The unit quantity of **Cell Volume** is **volume**.

Cell Volume Derivative (in the **Mesh...** category) is the change of a cell volume overtime.

Cell Volume Error (in the **Mesh...** category) is the cell volume over the unsteady cell volume.

2D Cell Volume

(in the **Mesh...** category) is the two-dimensional volume of a cell in an axisymmetric computation. For an axisymmetric computation, the 2D cell volume is scaled by the radius. Its unit quantity is **area**.

Cell Volume Change

(in the **Adaption...** category) is the maximum volume ratio of the current cell and its neighbors.

Cell Wall Distance

(in the **Mesh...** category) is the distribution of the normal distance of each cell centroid from the wall boundaries. Its unit quantity is **length**.

Cell Warpage

(in the **Adaption...** category) is the square root of the ratio of the distance between the cell centroid and cell circumcenter and the circumcenter radius:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{warpage} = \sqrt{\frac{\left|\vec{r}_{centroid} - \vec{r}_{circumcenter}\right|}{R_{circumcenter}}} \tag{33–6}$$

Cell Zone Index

(in the **Cell Info...** category) is the integer cell zone identification number. In problems that have more than one cell zone, the cell zone ID can be used to identify the various groups of cells.

Cell Zone Type

(in the **Cell Info...** category) is the integer cell zone type ID. A fluid cell has a type ID of 1, a solid cell has a type ID of 17, and an exterior cell (parallel solver) has a type ID of 21.

Compressibility Factor

(in the **Properties...** category) is the ratio of the ideal gas density of the fluid divided by the real gas fluid density in the in the same flow conditions. Compressibility Factor is defined as

$$Z = \frac{P/RT}{\rho} \tag{33–7}$$

where Z is the compressibility factor, P is the absolute pressure, T is the temperature, and $R = R_u/MW$ (the universal gas constant R_u divided by the molecular weight MW). The compressibility factor is available only with the real gas models.

Contact Resistivity

(in the **Solidification/Melting...** category) is the additional resistance at the wall due to contact resistance. It is equal to $R_c(1-\beta)/h$, where R_c is the contact resistance, β is the liquid fraction, and h is the cell height of the wall-adjacent cell. The unit quantity for **Contact Resistivity** is **thermal-resistivity**.

Critical Pressure

(in the **Properties...** category) is the mixture critical pressure. This property is available when a composition dependent option is selected for critical pressure in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Critical Specific Volume

(in the **Properties...** category) is the mixture critical specific volume. This property is available when a composition dependent option is selected for critical specific volume in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Critical Strain Rate

(in the **Premixed Combustion...** category) is a parameter that takes into account the stretching and extinction of premixed flames (g_{cr} in Equation 9–18 in the Theory Guide). Its unit quantity is **time-inverse**.

Critical Temperature

(in the **Properties...** category) is the mixture critical temperature. This property is available when a composition dependent option is selected for critical temperature in the cases with Aungier-Redlich-Kwong real gas model and species transport

Custom Field Functions...

are scalar field functions defined by you. You can create a custom function using the *Custom Field Function Calculator Dialog Box* (p. 2106). All defined custom field functions will be listed in the lower drop-down list. See *Custom Field Functions* (p. 1579) for details.

Damkohler Number

(in the **Premixed Combustion...** category) is a nondimensional parameter that is defined as the ratio of turbulent to chemical time scales.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Density...
includes variables related to density.

Density
(in the **Density...** category) is the mass per unit volume of the fluid. Plots or reports of **Density** include only fluid cell zones. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. The unit quantity for **Density** is **density**.

Density All
(in the **Density...** category) is the mass per unit volume of the fluid or solid material. Plots or reports of **Density All** include both fluid and solid cell zones. The unit quantity for **Density All** is **density**.

Derivatives...
are the viscous derivatives. For example, **dX-Velocity/dx** is the first derivative of the x component of velocity with respect to the x-coordinate direction. You can compute first derivatives of velocity, angular velocity, and pressure in the pressure-based solver, and first derivatives of velocity, angular velocity, temperature, and species in the density-based solvers.

Diameter
(in the **Properties...** category) is the diameter of particles, droplets, or bubbles of the secondary phase selected in the **Phase** drop-down list. Its unit quantity is **length**.

Diffusion Coef. of Scalar-n
(in the **User Defined Scalars...** category) is the diffusion coefficient for the nth user-defined scalar transport equation. See the UDF manual for details about defining user-defined scalars.

Discrete Phase Model...
includes quantities related to the discrete phase model. See *Modeling Discrete Phase* (p. 1007) for details about this model.

DPM Absorption Coefficient
(in the **Discrete Phase Model...** category) is the absorption coefficient for discrete-phase calculations that involve radiation (a in Equation 5–17 in the Theory Guide). Its unit quantity is **length-inverse**.

DPM Accretion
(in the **Discrete Phase Model...** category) is the accretion rate calculated at a wall boundary:

$$R_{accretion} = \sum_{p=1}^{N} \frac{\dot{m}_p}{A_{face}} \qquad \textbf{(33–8)}$$

where $\dot{m}_p$ is the mass flow rate of the particle stream, and A_{face} is the area of the wall face where the particle strikes the boundary. This item will appear only if the optional erosion/accretion model is enabled. See *Monitoring Erosion/Accretion of Particles at Walls* (p. 1017) for details. The unit quantity for **DPM Accretion** is **mass-flux**.

DPM Burnout
(in the **Discrete Phase Model...** category) is the exchange of mass from the discrete to the continuous phase for the combustion law (Law 5) and is proportional to the solid phase reaction rate. The burnout exchange has units of **mass-flow**.

DPM Concentration
(in the **Discrete Phase Model...** category) is the total concentration of the discrete phase. Its unit quantity is **density**.

DPM Emission
(in the **Discrete Phase Model...** category) is the amount of radiation emitted by a discrete-phase particle per unit volume. Its unit quantity is **heat-generation-rate**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM Enthalpy Source

(in the **Discrete Phase Model...** category) is the exchange of enthalpy (sensible enthalpy plus heat of formation) from the discrete phase to the continuous phase. The exchange is positive when the particles are a source of heat in the continuous phase. The unit quantity for **DPM Enthalpy Source** is **power**.

DPM Erosion

(in the **Discrete Phase Model...** category) is the erosion rate calculated at a wall boundary face:

$$R_{erosion} = \sum_{p=1}^{N} \frac{\dot{m}_p f(\alpha)}{A_{face}} \qquad \textbf{(33–9)}$$

where $\dot{m}_p$ is the mass flow rate of the particle stream, α is the impact angle of the particle path with the wall face, $f(\alpha)$ is the function specified in the *Wall Dialog Box* (p. 1865), and A_{face} is the area of the wall face where the particle strikes the boundary. This item will appear only if the optional erosion/accretion model is enabled. See *Monitoring Erosion/Accretion of Particles at Walls* (p. 1017) for details. The unit quantity for **DPM Erosion** is **mass-flux**.

DPM Evaporation/Devolatilization

(in the **Discrete Phase Model...** category) is the exchange of mass, due to droplet-particle evaporation or combusting-particle devolatilization, from the discrete phase to the evaporating or devolatilizing species. If you are not using the non-premixed combustion model, the mass source for each individual species (**DPM species-n Source**, below) is also available; for non-premixed combustion, only this sum is available. The unit quantity for **DPM Evaporation/Devolatilization** is **mass-flow**.

DPM Mass Source

(in the **Discrete Phase Model...** category) is the total exchange of mass from the discrete phase to the continuous phase. The mass exchange is positive when the particles are a source of mass in the continuous phase. If you are not using the non-premixed combustion model, **DPM Mass Source** will be equal to the sum of all species mass sources (**DPM species-n Source**, below); if you are using the non-premixed combustion model, it will be equal to **DPM Burnout** plus **DPM Evaporation/Devolatilization**. The unit quantity for **DPM Mass Source** is **mass-flow**.

DPM Scattering

(in the **Discrete Phase Model...** category) is the scattering coefficient for discrete-phase calculations that involve radiation (σ_s in Equation 5–17 in the Theory Guide). Its unit quantity is **length-inverse**.

DPM Sensible Enthalpy Source

(in the **Discrete Phase Model...** category) is the exchange of sensible enthalpy from the discrete phase to the continuous phase. The exchange is positive when the particles are a source of heat in the continuous phase. Its unit quantity is **power**.

DPM species-n Source

(in the **Discrete Phase Model...** category) is the exchange of mass, due to droplet-particle evaporation or combusting-particle devolatilization, from the discrete phase to the evaporating or devolatilizing species. (The name of the species will replace **species-n** in **DPM species-n Source**.) These species are specified in the *Set Injection Properties Dialog Box* (p. 2097), as described in *Defining Injection Properties* (p. 1040). The unit quantity is **mass-flow**. Note that this variable will not be available if you are using the non-premixed combustion model; use **DPM Evaporation/Devolatilization** instead.

DPM Swirl Momentum Source

(in the **Discrete Phase Model...** category) is the exchange of swirl momentum from the discrete phase to the continuous phase. This value is positive when the particles are a source of momentum in the continuous phase. The unit quantity is **force**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM X, Y, Z Momentum Source

(in the **Discrete Phase Model...** category) are the exchange of x-, y-, and z-direction momentum from the discrete phase to the continuous phase. These values are positive when the particles are a source of momentum in the continuous phase. The unit quantity is **force**.

Dynamic Cell Volume Change (in the **Mesh...** category) is the change of a cell volume.

Dynamic Pressure

(in the **Pressure...** category) is defined as $q \equiv \frac{1}{2}\rho v^2$. Its unit quantity is **pressure**.

Eff Diff Coef of species-n

(in the **Species...** category) is the sum of the laminar and turbulent diffusion coefficients of a species into the mixture:

$$D_{i,m} + \frac{\mu_t}{\rho Sc_t} \tag{33–10}$$

(The name of the species will replace **species-n** in **Eff Diff Coef of species-n**.) The unit quantity is **mass-diffusivity**.

Effective Prandtl Number

(in the **Turbulence...** category) is the ratio $\mu_{eff} c_p / k_{eff}$, where μ_{eff} is the effective viscosity, c_p is the specific heat, and k_{eff} is the effective thermal conductivity.

Effective Thermal Conductivity

(in the **Properties...** category) is the sum of the laminar and turbulent thermal conductivities, $k + k_t$, of the fluid. A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator). Its unit quantity is **thermal-conductivity**.

Effective Viscosity

(in the **Turbulence...** category) is the sum of the laminar and turbulent viscosities of the fluid. Viscosity, μ, is defined by the ratio of shear stress to the rate of shear. Its unit quantity is **viscosity**.

Enthalpy

(in the **Temperature...** category) is defined differently for compressible and incompressible flows, and depending on the solver and models in use.

For compressible flows,

$$H = \sum_j Y_j H_j \tag{33–11}$$

and for incompressible flows,

$$H = \sum_j Y_j H_j + \frac{p}{\rho} \tag{33–12}$$

where Y_j and H_j are, respectively, the mass fraction and enthalpy of species j. (See **Enthalpy of species-n**, below). For the pressure-based solver, the second term on the right-hand side of *Equation 33–12* (p. 1555) is included only if the pressure work term is included in the energy equation (see Inclusion of Pressure Work and Kinetic Energy Terms in the Theory Guide). For multiphase models, this value corresponds to

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the selected phase in the **Phase** drop-down list. For all reacting flow models, the **Enthalpy** plots consist of the thermal (or sensible) plus chemical energy. The unit quantity for **Enthalpy** is **specific-energy**.

In the case of the inert model (Using the Non-Premixed Model with the Inert Model in the Theory Guide), the enthalpy in a cell is split into the contributions from the inert and the reacting fractions of the gas phase species in the cell. The cell enthalpy is partitioned as

$$H = \gamma H_{inert} + (1-\gamma) H_{pdf} \tag{33–13}$$

where γ is the fraction of inert species in the cell. The quantity H_{inert} is the enthalpy of the inert species at the cell temperature, similarly H_{pdf} is the enthalpy of the active species at the cell temperature. It is assumed that the cell temperature is common to both inert and active species, so H_{inert}, H_{pdf} and the cell temperature are chosen so that *Equation 33–13* (p. 1556) is satisfied.

Enthalpy of species-n

(in the **Species...** category) is defined differently depending on the solver and models options in use. The quantity:

$$H_j = \int_{T_{ref,j}}^{T} c_{p,j} dT + h_j^0 \left(T_{ref,j} \right) \tag{33–14}$$

where $h_j^0 \left(T_{ref,j} \right)$ is the formation enthalpy of species j at the reference temperature $\left(T_{ref,j} \right)$, is reported only for non-adiabatic PDF cases, or if the density-based solver is selected. The quantity:

$$h_j = \int_{T_{ref}}^{T} c_{p,j} dT \tag{33–15}$$

where $T_{ref} = 298.15K$, is reported in all other cases. The unit quantity for **Enthalpy of species-n** is **specific-energy**.

Entropy

(in the **Temperature...** category) is a thermodynamic property defined by the equation

$$\Delta S \equiv \int_{rev} \frac{\delta Q}{T} \tag{33–16}$$

$$\Delta S = C_p \ln \left(\frac{T}{T_{ref}} \right) - R \ln \left(\frac{P}{P_{ref}} \right) \tag{33–17}$$

the entropy is computed using the equation

$$\Delta S = C_p \ln \left(\frac{T}{T_{ref}} \right) \tag{33–18}$$

The unit quantity for entropy is **specific-heat**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that for the real gas models the entropy is computed accordingly by the appropriate equation of state formulation.

Existing Value

(in the **Adaption...** category) is the value that presently resides in the temporary space reserved for cell variables (i.e., the last value that you displayed or computed).

Face Area Magnitude

(in the **Mesh...** category) is the magnitude of the face area vector for noninternal faces (i.e., faces that only have `c0` and no `c1`). The values are stored on the face itself and used when required. This variable is intended only for zone surfaces and not for other surfaces created for postprocessing.

Face Handedness

(in the **Mesh...** category) is a parameter that is equal to one in cells that are adjacent to left-handed faces, and zero elsewhere. It can be used to locate mesh problems.

Fine Scale Mass Fraction of species-n

(in the **Species...** category) is the term Y_i^* in Equation 7–29 in the Theory Guide.

Fine Scale Temperature

(in the **Temperature...** category) is the temperature of the fine scales, which is calculated from the enthalpy when the reaction proceeds over the time scale (τ^* in Equation 7–28 in the Theory Guide), governed by the Kinetic rates of Equation 7–8 in the Theory Guide. Its unit quantity is **temperature**.

Fine Scale Transfer Rate

(in the **Species...** category) is the transfer rate of the fine scales, which is equal to the inverse of the time scale (τ^* in Equation 7–28 in the Theory Guide). Its unit quantity is **time-inverse**.

1-Fine Scale Volume Fraction

(in the **Species...** category) is a function of the fine scale volume fraction (ξ^* in Equation 7–27 in the Theory Guide). The quantity is subtracted from unity to make it easier to interpret.

Fvar Prod

(in the **Pdf...** category) is the production term in the mixture fraction variance equation solved in the non-premixed combustion model (i.e., the last two terms in Equation 8–5 in the Theory Guide).

Fvar2 Prod

(in the **Pdf...** category) is the production term in the secondary mixture fraction variance equation solved in the non-premixed combustion model. See Equation 8–5 in the Theory Guide.

Gas Constant (R)

(in the **Properties...** category) is the gas constant of the fluid. Its unit quantity is **specific-heat**.

Granular Conductivity

(in the **Properties...** category) is equivalent to the diffusion coefficient in Equation 17–211 in the Theory Guide. For more information, see Granular Temperature in the Theory Guide. Its unit quantity is kg/m-s.

Granular Pressure...

includes quantities for reporting the solids pressure for each granular phase (p_s in Equation 17–181 in the Theory Guide). See Solids Pressure in the Theory Guide for details. Its unit quantity is **pressure**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Temperature...

includes quantities for reporting the granular temperature for each granular phase (Θ_s in Equation 17–211 in the Theory Guide). See Granular Temperature in the Theory Guide for details. Its unit quantity is m^2/s^2. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Mesh...

includes variables related to the mesh.

Mesh X-Velocity, Mesh Y-Velocity, Mesh Z-Velocity

(in the **Velocity...** category) are the vector components of the mesh velocity for moving-mesh problems (rotating or multiple reference frames, mixing planes, or sliding meshes). Its unit quantity is **velocity**.

HCN Density

(in the **NOx...** category) is the mass per unit volume of HCN. The unit quantity is **density**. The **HCN Density** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Heat of Heterogeneous Reaction

(in the **Phase Interaction...** category) is the heat added or removed due to heterogeneous chemical reactions. For exothermic reactions the **Heat of Heterogeneous Reaction** is reported as a positive quantity, while for endothermic reactions it will be a negative quantity. If you have more than one heterogeneous reaction defined in your case, the **Heat of Heterogeneous Reaction** reported is the sum of the heat for all heterogeneous reactions. The unit quantity of **Heat of Heterogeneous Reaction** is Watt.

Heat of Reaction

(in the **Reactions...** category) is the heat added or removed due to chemical reactions, as defined in Equation 5–10 in the Theory Guide. For exothermic reactions, the heat of reaction is reported as a positive quantity, while for endothermic reactions it is reported as a negative quantity. If you have more than one reaction defined in your case, the **Heat of Reaction** reported is the sum of the heat for all reactions. The unit of measurement for the heat of reaction is Watts. The **Heat of Reaction** is not available for the non-premixed and partially-premixed models.

Helicity

(in the **Velocity...** category) is defined by the dot product of vorticity and the velocity vector.

$$H = (\nabla \times \vec{V}) \cdot \vec{V} \qquad \textbf{(33–19)}$$

Vorticity is a measure of the rotation of a fluid element as it moves in the flow field.

Incident Radiation

(in the **Radiation...** category) is the total radiation energy, G, that arrives at a location per unit time and per unit area:

$$G = \int_{\Omega=4\pi} I d\Omega \qquad \textbf{(33–20)}$$

where I is the radiation intensity and Ω is the solid angle. G is the quantity that the P-1 radiation model computes. For the DO radiation model, the incident radiation is computed over a finite number of discrete solid angles, each associated with a vector direction. The unit quantity for **Incident Radiation** is **heat-flux**.

Incident Radiation (Band n)

(in the **Radiation...** category) is the radiation energy contained in the wavelength band $\Delta\lambda$ for the non-gray P-1 radiation model or the non-gray DO radiation model. Its unit quantity is **heat-flux**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Intermittency Factor (γ)

(in the **Turbulence...** category) is a measure of the probability that a given point is located inside a turbulent region. Upstream of transition the intermittency is zero. Once the transition occurs, the intermittency is ramped up to one until the fully turbulent boundary layer regime is achieved.

Internal Energy

(in the **Temperature...** category) is the summation of the kinetic and potential energies of the molecules of the substance per unit volume (and excludes chemical and nuclear energies). **Internal Energy** is defined as $e = c_v T$. Its unit quantity is **specific-energy**.

Jet Acoustic Power

(in the **Acoustics...** category) is the acoustic power for turbulent axisymmetric jets (see Equation 15–14 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Jet Acoustic Power Level (dB)

(in the **Acoustics...** category) is the acoustic power for turbulent axisymmetric jets, reported in dB (see Equation 15–27 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Kinetic Rate of Reaction-n

(in the **Reactions...** category) is given by the following expression (see Equation 7–8 in the Theory Guide for definitions of the variables shown here):

$$\hat{R}_r = \Gamma \left(k_{f,r} \prod_{j=1}^{N_r} \left[C_{j,r} \right]^{\eta'_{j,r}} - k_{b,r} \prod_{j=1}^{N_r} \left[C_{j,r} \right]^{\eta''_{j,r}} \right) \qquad \textbf{(33–21)}$$

The reported value is independent of any particular species, and has units of kmol/m^3-s.

To find the rate of production/destruction for a given species i due to reaction r, multiply the reported reaction rate for reaction r by the term $M_i \left(v''_{i,r} - v'_{i,r} \right)$, where M_i is the molecular weight of species i, and $v''_{i,r}$ and $v'_{i,r}$ are the stoichiometric coefficients of species i in reaction r.

For particle reactions it is the global rate of the particle reaction n expressed in kmol/s/m3. This is computed as

$$\frac{\overline{R}_{j,r}}{M_j V}$$

where $\overline{R}_{j,r}$ is the rate of particle species depletion (or generation) given by Equation 7–66 in the Theory Guide, M_j is the particle species molecular weight, and V is the cell volume.

Lam Diff Coef of species-n

(in the **Species...** category) is the laminar diffusion coefficient of a species into the mixture, $D_{i,m}$. Its unit quantity is **mass-diffusivity**.

Laminar Flame Speed

(in the **Premixed Combustion...** category) is the propagation speed of laminar premixed flames (U_l in Equation 9–9 in the Theory Guide). Its unit quantity is **velocity**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Laminar Kinetic Energy (kl)

(in the **Turbulence...category**) is a measure of the "laminar" streamwise fluctuations present in the pre-transitional region of the boundary layer subjected to free-stream turbulence. A transport equation of kl is considered by the k-kl-omega transition model.

LEE Self-Noise X-Source, LEE Self-Noise Y-Source, LEE Self-Noise Z-Source

(in the **Acoustics...** category) are the self-noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 15–32 in the Theory Guide). They are available only when the **Broadband Noise Sources** acoustics model is being used.

LEE Shear-Noise X-Source, LEE Shear-Noise Y-Source, LEE Shear-Noise Z-Source

(in the **Acoustics...** category) are the shear-noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 15–32 in the Theory Guide). They are available only when the **Broadband Noise Sources** acoustics model is being used.

LEE Total Noise X-Source, LEE Total Noise Y-Source, LEE Total Noise Z-Source

(in the **Acoustics...** category) are the total noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 15–32 in the Theory Guide). The total noise source term is the sum of the self-noise and shear-noise source terms. They are available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Self-Noise Source

(in the **Acoustics...** category) is the self-noise source term in the linearized Lilley's equation (see Equation 15–36 in the Theory Guide), available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Shear-Noise Source

(in the **Acoustics...** category) is the shear-noise source term in the linearized Lilley's equation (see Equation 15–36 in the Theory Guide), available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Total Noise Source

(in the **Acoustics...** category) is the total noise source term in the linearized Lilley's equation (see Equation 15–36 in the Theory Guide). The total noise source term is the sum of the self-noise and shear-noise source terms, available only when the **Broadband Noise Sources** acoustics model is being used.

Liquid Fraction

(in the **Solidification/Melting...** category) is the liquid fraction β computed by the solidification/melting model:

$$\beta = \frac{\Delta H}{L} = 0 \quad if \;\; T < T_{solidus} \tag{33–22}$$

$$\beta = \frac{\Delta H}{L} = 1 \quad if \;\; T > T_{liquidus} \tag{33–23}$$

$$\beta = \frac{\Delta H}{L} = \frac{T - T_{solidus}}{T_{liquidus} - T_{solidus}} \quad if \;\; T_{solidus} < T < T_{liquidus} \tag{33–24}$$

Mach Number

(in the **Velocity...** category) is the ratio of velocity and speed of sound.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass fraction of HCN, Mass fraction of NH3, Mass fraction of NO, Mass fraction of N2O

(in the **NOx...** category) are the mass of HCN, the mass of NH_3, the mass of NO, and the mass of N_2O per unit mass of the mixture (e.g., kg of HCN in 1 kg of the mixture). The **Mass fraction of HCN** and the **Mass fraction of NH3** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Mass fraction of nuclei

(in the **Soot...** category) is the number of particles per unit mass of the mixture (in units of particles $\times 10^{15}$/kg) The **Mass fraction of nuclei** will appear only if you use the two-step soot model. See *Soot Formation* (p. 973) for details.

Mass fraction of soot

(in the **Soot...** category) is the mass of soot per unit mass of the mixture (e.g., kg of soot in 1 kg of the mixture). See *Soot Formation* (p. 973) for details.

Mass fraction of species-n

(in the **Species...** category) is the mass of a species per unit mass of the mixture (e.g., kg of species in 1 kg of the mixture).

Mean quantity-n

(in the **Unsteady Statistics...** category) is the time-averaged value of a solution variable (e.g., **Static Pressure**). See *Postprocessing for Time-Dependent Problems* (p. 1267) for details.

Meridional Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate that follows the flow path from inlet to outlet. Its value varies from 0 to 1.

Mixture Fraction Variance

(in the **Pdf...** category) is the variance of the mixture fraction solved for in the non-premixed combustion model. This is the second conservation equation (along with the mixture fraction equation) that the non-premixed combustion model solves. (See Definition of the Mixture Fraction in the Theory Guide.)

Modified Turbulent Viscosity

(in the **Turbulence...** category) is the transported quantity $\tilde{\nu}$ that is solved for in the Spalart-Allmaras turbulence model (see Equation 4–15 in the Theory Guide). The turbulent viscosity, μ_t, is computed directly from this quantity using the relationship given by Equation 4–16 in the Theory Guide. Its unit quantity is **viscosity**.

Molar Concentration of species-n

(in the **Species...** category) is the moles per unit volume of a species. Its unit quantity is **concentration**.

Mole fraction of species-n

(in the **Species...** category) is the number of moles of a species in one mole of the mixture.

Mole fraction of HCN, Mole fraction of NH3, Mole fraction of NO, Mole fraction of N2O

(in the **NOx...** category) are the number of moles of HCN, NH_3, NO, and N_2O in one mole of the mixture. The **Mole fraction of HCN** and the **Mole fraction of NH3** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Mole fraction of soot

(in the **Soot...** category) is the number of moles of soot in one mole of the mixture.

Molecular Prandtl Number

(in the **Properties...** category) is the ratio $c_p\mu_{lam}/k_{lam}$.

Molecular Viscosity

(in the **Properties...** category) is the laminar viscosity of the fluid. Viscosity, μ, is defined by the ratio of shear stress to the rate of shear. Its unit quantity is **viscosity**. For multiphase models, this value corres-

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ponds to the selected phase in the **Phase** drop-down list. For granular phases, this is equivalent to the solids shear viscosity μ_s in Equation 17–198 in the Theory Guide.

Momentum Thickness Re ($Re_{\theta t}$)

(in the **Turbulence...** category) is based on the momentum thickness of the boundary layer. The SST transition model is considering a non local empirical correlation for the value of $Re_{\theta t}$ in the free-stream, based on turbulence intensity, pressure gradient, etc...and a transport equation to allow the free-stream value to diffuse into the boundary layer.

NH3 Density, NO Density, N2O Density

(in the **NOx...** category) are the mass per unit volume of NH_3, NO and N_2O. The unit quantity for each is **density**. The **NH3 Density** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

NOx...

contains quantities related to the NOx model. See *NOx Formation* (p. 943) for details about this model.

Orthogonal Quality

(in the **Mesh...** category) is a measure of the quality of a mesh, and is computed for each cell using the vectors from the cell centroid to each of its faces, the area vectors of the cell's faces, and the vectors from the cell centroid to the centroids of each of the adjacent cells (see *Equation 6–1* (p. 144) and *Equation 6–2* (p. 144)). The worst cells will have an **Orthogonal Quality** closer to 0, with better cells closer to 1.

Partition Boundary Cell Distance

(in the **Mesh...** category) is the smallest number of cells which must be traversed to reach the nearest partition (interface) boundary.

Partition Neighbors

(in the **Cell Info...** category) is the number of adjacent partitions (i.e., those that share at least one partition boundary face (interface)). It gives a measure of the number of messages that will have to be generated for parallel processing.

Pdf...

contains quantities related to the non-premixed combustion model, which is described in *Modeling Non-Premixed Combustion* (p. 839).

PDF Table Adiabatic Enthalpy

is the adiabatic enthalpy corresponding to the cell value of mixture fraction. For single mixture fraction cases it is given by the following equation:

$$H_{ad} = H_{fuel} f + H_{oxidizer}(1 - f) \tag{33–25}$$

and for cases involving a secondary stream it is given by the following equation:

$$H_{ad} = H_{fuel} f + H_{secondary} f_{sec} + H_{oxidizer}\left(1 - f_{sec} - f\right) \tag{33–26}$$

where

f = mixture fraction

f_{sec} = secondary mixture fraction

H_{fuel} = total enthalpy of the fuel stream

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$H_{secondary}$ = total enthalpy of the secondary stream

$H_{oxidizer}$ = total enthalpy of the oxidizer stream

For adiabatic cases the **PDF Table Adiabatic Enthalpy** is equal to the value of Enthalpy. The unit of measurement is specific-energy.

PDF Table Heat Loss/Gain

is given by the following equation:

$$h_{loss} = (H - H_{min}) / (H_{ad} - H_{min}) - 1 \quad \textbf{(33–27)}$$

if the cell enthalpy is less than the adiabatic enthalpy, and by the following equation:

$$h_{gain} = 1 - (H_{max} - H) / (H_{max} - H_{ad}) \quad \textbf{(33–28)}$$

if the cell enthalpy is higher than adiabatic

where

H = total enthalpy

H_{ad} = the PDF Table Adiabatic Enthalpy

H_{min} = the minimum Enthalpy defined in the PDF table

H_{max} = the maximum Enthalpy defined in the PDF table

The **PDF Table Heat Loss/Gain** is dimensionless and ranges in value from -1, when H is equal to H_{min}, to +1, when H is equal to H_{max}. If H is equal to the adiabatic enthalpy it will be 0.

Phases...

contains quantities for reporting the volume fraction of each phase. See *Modeling Multiphase Flows* (p. 1091) for details.

Pitchwise Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate in the circumferential (pitchwise) direction. Its value varies from 0 to 1.

Preconditioning Reference Velocity

(in the **Velocity...** category) is the reference velocity used in the coupled solver's preconditioning algorithm. See Preconditioning in the Theory Guide for details.

Premixed Combustion...

contains quantities related to the premixed combustion model, which is described in *Modeling Premixed Combustion* (p. 893).

Pressure...

includes quantities related to a normal force per unit area (the impact of the gas molecules on the surfaces of a control volume).

Pressure Coefficient

(in the **Pressure...** category) is a dimensionless parameter defined by the equation

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$C_p = \frac{\left(p - p_{ref}\right)}{q_{ref}} \tag{33–29}$$

and q_{ref} is the reference dynamic pressure defined by $\frac{1}{2}\rho_{ref}v_{ref}^{2}$. The reference pressure, density, and velocity are defined in the *Reference Values Task Page* (p. 1898).

Product Formation Rate

(in the **Premixed Combustion...** category) is the source term in the progress variable transport equation (S_c in Equation 9–1 in the Theory Guide). Its unit quantity is **time-inverse**.

Production of k

(in the **Turbulence...** category) is the rate of production of turbulence kinetic energy (times density). Its unit quantity is **turb-kinetic-energy-production**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Progress Variable

(in the **Premixed Combustion...** category) is a normalized mass fraction of the combustion products ($c = 1$) or unburnt mixture products ($c = 0$), as defined by Equation 9–2 in the Theory Guide.

Properties...

includes material property quantities for fluids and solids.

Rate of NO

(in the **NOx...** category) is the overall rate of formation of NO due to all active NO formation pathways (e.g., thermal, prompt, etc.).

Rate of Nuclei

(in the **Soot...** category) is the overall rate of formation of nuclei.

Rate of N2OPath NO

(in the **NOx...** category) is the rate of formation of NO due to the N2O pathway only (only available when N2O pathway is active).

Rate of Prompt NO

(in the **NOx...** category) is the rate of formation of NO due to the prompt pathway only (only available when prompt pathway is active).

Rate of Reburn NO

(in the **NOx...** category) is the rate of formation of NO due to the reburn pathway only (only available when reburn pathway is active).

Rate of SNCR NO

(in the **NOx...** category) is the rate of formation of NO due to the SNCR pathway only (only available when SNCR pathway is active).

Rate of Soot

(in the **Soot...** category) is the overall rate of formation of soot mass.

Rate of Thermal NO

(in the **NOx...** category) is the rate of formation of NO due to the thermal pathway only (only available when thermal pathway is active).

Rate of Fuel NO

(in the **NOx...** category) is the rate of formation of NO due to the fuel pathway only (only available when fuel pathway is active).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rate of USER NO

(in the **NOx...** category) is the rate of formation of NO due to user defined rates only (only available when UDF rates are added).

Radial Coordinate

(in the **Mesh...** category) is the length of the radius vector in the polar coordinate system. The radius vector is defined by a line segment between the node and the axis of rotation. You can define the rotational axis in the *Fluid Dialog Box* (p. 1796). (See also *Velocity Reporting Options* (p. 1531).) The unit quantity for **Radial Coordinate** is **length**.

Radial Pull Velocity

(in the **Solidification/Melting...** category) is the radial-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Radial Velocity

(in the **Velocity...** category) is the component of velocity in the radial direction. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for **Radial Velocity** is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Radial-Wall Shear Stress

(in the **Wall Fluxes...** category) is the radial component of the force acting tangential to the surface due to friction. Its unit quantity is **pressure**.

Radiation...

includes quantities related to radiation heat transfer. See *Modeling Radiation* (p. 704) for details about the radiation models available in ANSYS FLUENT.

Radiation Heat Flux

(in the **Wall Fluxes...** category) is the rate of radiation heat transfer through the control surface. It is calculated by the solver according to the specified radiation model. Heat flux out of the domain is negative, and heat flux into the domain is positive. The unit quantity for **Radiation Heat Flux** is **heat-flux**.

Radiation Temperature

(in the **Radiation...** category) is the quantity θ_R, defined by

$$\theta_R = \left(\frac{G}{4\sigma}\right)^{1/4} \tag{33–30}$$

where G is the **Incident Radiation**. The unit quantity for **Radiation Temperature** is **temperature**.

Rate of Reaction-n

(in the **Reactions...** category) is the effective rate of progress of nth reaction. For the finite-rate model, the value is the same as the **Kinetic Rate of Reaction-n**. For the eddy-dissipation model, the value is equivalent to the **Turbulent Rate of Reaction-n**. For the finite-rate/eddy-dissipation model, it is the lesser of the two.

For particle reactions it is the global rate of the particle reaction n expressed in kmol/s/m3. This is computed as

$$\frac{\overline{R}_{j,r}}{M_j V}$$

where $\overline{R}_{j,r}$ is the rate of particle species depletion (or generation) given by Equation 7–66 in the Theory Guide, M_j is the particle species molecular weight, and V is the cell volume.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reactions...

includes quantities related to finite-rate reactions. See *Modeling Species Transport and Finite-Rate Chemistry* (p. 799) for information about modeling finite-rate reactions.

Reduced Temperature

(in the **Properties...** category) is the ratio T/T_c of the fluid temperature T divided by the critical temperature T_c. The reduced temperature T_r is available only with the Angier-Redlich-Kwong real gas model.

Reduced Pressure

(in the **Properties...** category) is the ratio P/P_c of the fluid pressure P divided by the critical pressure P_c. The reduced pressure P_r is available only with the Angier-Redlich-Kwong real gas model.

Reflected Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux reflected by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Reflected Visible Solar Flux, Reflected IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux reflected by a semi-transparent wall or porous jump boundary for a visible or infrared (IR) radiation.

Refractive Index

(in the **Radiation...** category) is a nondimensional parameter defined as the ratio of the speed of light in a vacuum to that in a material. See Specular Semi-Transparent Walls in the Theory Guide for details.

Relative Axial Velocity

(in the **Velocity...** category) is the axial-direction component of the velocity relative to the reference frame motion. See *Velocity Reporting Options* (p. 1531) for details. The unit quantity for **Relative Axial Velocity** is **velocity**.

Relative Humidity

(in the **Species...** category) is the ratio of the partial pressure of the water vapor actually present in an air-water mixture to the saturation pressure of water vapor at the mixture temperature. ANSYS FLUENT computes the saturation pressure, p, from the following equation [*69*] (p. 2212):

$$\ln\left(\frac{p}{p_c}\right)=\left(\frac{T_c}{T}-1\right)\times\sum_{i=1}^{8}F_i\left[a\left(T-T_p\right)\right]^{i-1} \qquad \textbf{(33–31)}$$

where p_c = 22.089 MPa

T_c = 647.286 K

$F_1 = -7.4192420$

$F_2 = 2.9721000\times10^{-1}$

$F_3 = -1.1552860\times10^{-1}$

$F_4 = 8.6856350\times10^{-3}$

$F_5 = 1.0940980\times10^{-3}$

$F_6 = -4.3999300\times10^{-3}$

$F_7 = 2.5206580\times10^{-3}$

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$F_8 = -5.2186840 \times 10^{-4}$

$a = 0.01$

$T_p = 338.15$ K

Relative Length Scale (DES)

(in the **Turbulence...** category) is defined by

$$Ls = Ls_{rans} - Ls_{les} \quad \textbf{(33–32)}$$

where Ls_{rans} is an RANS-based length scale, and Ls_{les} is an LES-based length scale. All of the cells inside the domain in which $Ls > 0$ belong to the LES region, and all of the cells inside the domain in which $Ls < 0$ belong to the RANS region. If the **Delayed DES** option is enabled (default option), the relative length scale is defined by:

$$L = Ls_{rans} \times F - L_{les} \quad \textbf{(33–33)}$$

where F is based on the delaying function considered by the DES model ($F = Fd$ for the DES-SA model and the DES-RKE model and $F = (1 - Fsst)$ for the DES-SST model). It is equal to zero inside the boundary layer and equal to one outside.

Relative Mach Number

(in the **Velocity...** category) is the nondimensional ratio of the relative velocity and speed of sound.

Relative Radial Velocity

(in the **Velocity...** category) is the radial-direction component of the velocity relative to the reference frame motion. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for **Relative Radial Velocity** is **velocity**.

Relative Swirl Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity relative to the reference frame motion, in an axisymmetric swirling flow. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for **Relative Swirl Velocity** is **velocity**.

Relative Tangential Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity relative to the reference frame motion. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for **Relative Tangential Velocity** is **velocity**.

Relative Total Pressure

(in the **Pressure...** category) is the stagnation pressure computed using relative velocities instead of absolute velocities; i.e., for incompressible flows the dynamic pressure would be computed using the relative velocities. (See *Velocity Reporting Options* (p. 1531) for more information about relative velocities.) The unit quantity for **Relative Total Pressure** is **pressure**.

Relative Total Temperature

(in the **Temperature...** category) is the stagnation temperature computed using relative velocities instead of absolute velocities. (See *Velocity Reporting Options* (p. 1531) for more information about relative velocities.) The unit quantity for **Relative Total Temperature** is **temperature**.

Relative Velocity Angle

(in the **Velocity...** category) is similar to the **Velocity Angle** except that it uses the relative tangential velocity, and is defined as

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tan^{-1}\left(-\frac{\text{relative-tangential-velocity}}{\text{axial-velocity}}\right) \quad \textbf{(33–34)}$$

Its unit quantity is **angle**.

Relative Velocity Magnitude

(in the **Velocity...** category) is the magnitude of the relative velocity vector instead of the absolute velocity vector. The relative velocity ($\vec{w}$) is the difference between the absolute velocity ($\vec{v}$) and the mesh velocity. For simple rotation, the relative velocity is defined as

$$\vec{w} \equiv \vec{v} - \vec{\Omega} \times \vec{r} \quad \textbf{(33–35)}$$

where $\vec{\Omega}$ is the angular velocity of a moving reference frame about the origin and $\vec{r}$ is the position vector. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for **Relative Velocity Magnitude** is **velocity**.

Relative X Velocity, Relative Y Velocity, Relative Z Velocity

(in the **Velocity...** category) are the x-, y-, and z-direction components of the velocity relative to the reference frame motion. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for these variables is **velocity**.

Residuals...

contains different quantities for the pressure-based and density-based solvers:

In the density-based solvers, this category includes the corrections to the primitive variables pressure, velocity, temperature, and species, as well as the time rate of change of the corrections to these primitive variables for the current iteration (i.e., residuals). Corrections are the changes in the variables between the current and previous iterations and residuals are computed by dividing a cell's correction by its physical time step. The total residual for each variable is the summation of the Euler, viscous, and dissipation contributions. The dissipation components are the vector components of the flux-like, face-based dissipation operator.

In the pressure-based solver, only the **Mass Imbalance** in each cell is reported (unless you have requested others, as described in *Postprocessing Residual Values* (p. 1277)). At convergence, this quantity should be small compared to the average mass flow rate.

RMS quantity-n

(in the **Unsteady Statistics...** category) is the root mean squared value of a solution variable (e.g., **Static Pressure**). See *Postprocessing for Time-Dependent Problems* (p. 1267) for details.

Rothalpy

(in the **Temperature...** category) is defined as

$$I = h + \frac{w^2}{2} - \frac{u^2}{2} \quad \textbf{(33–36)}$$

where h is the enthalpy, w is the relative velocity magnitude, and u is the magnitude of the rotational velocity $\vec{u} = \vec{\omega} \times \vec{r}$.

Scalar-n

(in the **User Defined Scalars...** category) is the value of the nth scalar quantity you have defined as a user-defined scalar. See the UDF manual for more information about user-defined scalars.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Dissipation

(in the **Pdf...** category) is one of two parameters that describes the species mass fraction and temperature for a laminar flamelet in mixture fraction spaces. It is defined as

$$\chi = 2D|\nabla f|^2 \tag{33–37}$$

where f is the mixture fraction and D is a representative diffusion coefficient (see The Flamelet Concept in the Theory Guide for details). Its unit quantity is **time-inverse**.

Scattering Coefficient

(in the **Radiation...** category) is the property of a medium that describes the amount of scattering of thermal radiation per unit path length for propagation in the medium. It can be interpreted as the inverse of the mean free path that a photon will travel before undergoing scattering (if the scattering coefficient does not vary along the path). The unit quantity for **Scattering Coefficient** is **length-inverse**.

Secondary Mean Mixture Fraction

(in the **Pdf...** category) is the mean ratio of the secondary stream mass fraction to the sum of the fuel, secondary stream, and oxidant mass fractions. It is the secondary-stream conserved scalar that is calculated by the non-premixed combustion model. See Definition of the Mixture Fraction in the Theory Guide.

Secondary Mixture Fraction Variance

(in the **Pdf...** category) is the variance of the secondary stream mixture fraction that is solved for in the non-premixed combustion model. See Definition of the Mixture Fraction in the Theory Guide.

Sensible Enthalpy

(in the **Temperature...** category) is available when any of the species models are active and displays only the thermal (sensible) enthalpy.

Skin Friction Coefficient

(in the **Wall Fluxes...** category) is a nondimensional parameter defined as the ratio of the wall shear stress and the reference dynamic pressure

$$C_f \equiv \frac{\tau_w}{\frac{1}{2}\rho_{ref} v_{ref}^2} \tag{33–38}$$

v_{ref} are the reference density and velocity defined in the *Reference Values Task Page* (p. 1898). For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Solar Heat Flux

(in the **Wall Fluxes...** category) is the rate of solar heat transfer through the control surface. Heat flux out of the domain is negative and heat flux into the domain is positive.

Solidification/Melting...

contains quantities related to solidification and melting.

Soot...

contains quantities related to the **Soot** model, which is described in *Soot Formation* (p. 973).

Soot Density

(in the **Soot...** category) is the mass per unit volume of soot. The unit quantity is **density**. See Fuel NOx Formation in the Theory Guide for details.

Sound Speed

(in the **Properties...** category) is the acoustic speed. It is computed from $\sqrt{\frac{\gamma p}{\rho}}$. Its unit quantity is **velocity**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that for the real gas models the sound speed is computed accordingly by the appropriate equation of state formulation.

Spanwise Coordinate
(in the **Mesh...** category) is the normalized (dimensionless) coordinate in the spanwise direction, from hub to casing. Its value varies from 0 to 1.

species-n Source Term
(in the **Species...** category) is the source term in each of the species transport equations due to reactions. The unit quantity is always kg/m ^{3}-s.

Species...
includes quantities related to species transport and reactions.

Specific Dissipation Rate (Omega)
(in the **Turbulence...** category) is the rate of dissipation of turbulence kinetic energy in unit volume and time. Its unit quantity is **time-inverse**.

Specific Heat (Cp)
(in the **Properties...** category) is the thermodynamic property of specific heat at constant pressure. It is defined as the rate of change of enthalpy with temperature while pressure is held constant. Its unit quantity is **specific-heat**.

Specific Heat Ratio (gamma)
(in the **Properties...** category) is the ratio of specific heat at constant pressure to the specific heat at constant volume.

Spinodal Temperature
(in the **Properties...** category) is the temperature at which the derivative of pressure with respect to volume becomes positive. The **spinodal temperature** defines the point beyond which the equation of state is no longer valid for the gas phase. If the temperature of your case approaches the **spinodal temperature** in some regions, this indicates that the flow conditions in these regions may fall inside the saturation dome. The **spinodal temperature** is available only with the Angier-Redlich-Kwong real gas model.

Stored Cell Partition
(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of ANSYS FLUENT, the partition ID can be used to determine the extent of the various groups of cells. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. See *Partitioning the Mesh Manually and Balancing the Load* (p. 1605) for more information.

Static Pressure
(in the **Pressure...** category) is the static pressure of the fluid. It is a gauge pressure expressed relative to the prescribed operating pressure. The absolute pressure is the sum of the **Static Pressure** and the operating pressure. Its unit quantity is **pressure**.

Static Temperature
(in the **Temperature...**and **Premixed Combustion...** categories) is the temperature that is measured moving with the fluid. Its unit quantity is **temperature**.

Note that **Static Temperature** will appear in the **Premixed Combustion...** category only for adiabatic premixed combustion calculations. See *Postprocessing for Premixed Combustion Calculations* (p. 901).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Strain Rate

(in the **Derivatives...** category) relates shear stress to the viscosity. Also called the shear rate ($\dot{\gamma}$ in *Equation 8–29* (p. 414)), the strain rate is related to the second invariant of the rate-of-deformation tensor $\overline{\overline{D}}$. Its unit quantity is **time-inverse**. In 3D Cartesian coordinates, the strain rate, S, is defined as

$$\begin{aligned} S^2 = & \left[\frac{\partial u}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial x}\right)+\frac{\partial u}{\partial y}\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)+\frac{\partial u}{\partial z}\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+ \\ & \left[\frac{\partial v}{\partial x}\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)+\frac{\partial v}{\partial y}\left(\frac{\partial v}{\partial y}+\frac{\partial v}{\partial y}\right)+\frac{\partial v}{\partial z}\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+ \\ & \left[\frac{\partial w}{\partial x}\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)+\frac{\partial w}{\partial y}\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)+\frac{\partial w}{\partial z}\left(\frac{\partial w}{\partial z}+\frac{\partial w}{\partial z}\right)\right] \end{aligned} \tag{33–39}$$

For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Stream Function

(in the **Velocity...** category) is formulated as a relation between the streamlines and the statement of conservation of mass. A streamline is a line that is tangent to the velocity vector of the flowing fluid. For a 2D planar flow, the stream function, ψ, is defined such that

$$\rho u \equiv \frac{\partial \psi}{\partial y} \rho v \equiv -\frac{\partial \psi}{\partial x} \tag{33–40}$$

constant values of stream function defining two streamlines is the mass rate of flow between the streamlines.

The accuracy of the stream function calculation is determined by the text command `/display/set/n-stream-func`.

Stretch Factor

(in the **Premixed Combustion...** category) is a nondimensional parameter that is defined as the probability of unquenched flamelets (G in Equation 9–15 in the Theory Guide).

Subgrid Dissipation Rate

(in the **Turbulence...** category) is the turbulence dissipation rate of the unresolved eddies, eps_{sgs}, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$eps_{sgs} = C_{eps} * \frac{k_{sgs}^{3/2}}{L_s} \tag{33–41}$$

Its unit quantity is **turbulent-energy-diss-rate**.

Subgrid Filter Length

(in the **Turbulence...** category) is a mixing length for subgrid scales of the LES turbulence model (defined as L_S in Equation 4–252 in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subgrid Kinetic Energy

(in the **Turbulence...** category) is the turbulence kinetic energy per unit mass of the unresolved eddies, k_{sgs}, calculated using a transport equation, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$k_{sgs}=\frac{1}{2}\left(\overline{u_k^2}-\overline{u}_k^2\right) \quad \textbf{(33–42)}$$

Its unit quantity is **turbulent-kinetic-energy**.

Subgrid Turbulent Viscosity

(in the **Turbulence...** category) is the turbulent (dynamic) viscosity of the fluid calculated using the LES turbulence model. It expresses the proportionality between the anisotropic part of the subgrid-scale stress tensor and the rate-of-strain tensor. (See Equation 4–243 in the Theory Guide.) Its unit quantity is **viscosity**.

Subgrid Turbulent Viscosity Ratio

(in the **Turbulence...** category) is the ratio of the subgrid turbulent viscosity of the fluid to the laminar viscosity, calculated using the LES turbulence model.

Subtest Kinetic Energy

(in the **Turbulence...** category) is the turbulence kinetic energy of filtered eddies, kf, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$k_f=L_1+L_2+L_3 \quad \textbf{(33–43)}$$

with L_i being the normal components of the Leonard stress.

Its unit quantity is **turbulent-kinetic-energy**

Surface Acoustic Power

(in the **Acoustics...** category) is the **Acoustic Power** per unit area generated by boundary layer turbulence (see Equation 15–31 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used. Its unit quantity is **power** per **area**.

Surface Acoustic Power Level (dB)

(in the **Acoustics...** category) is the **Acoustic Power** per unit area generated by boundary layer turbulence, and represented in dB (see Equation 15–31 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Surface Cluster ID

(in the **Radiation...** category) is used to view the distribution of surface clusters in the domain. Each cluster has a unique integer number (ID) associated with it.

Surface Coverage of species-n

(in the **Species...** category) is the amount of a surface species that is deposited on the substrate at a specific point in time.

Surface Deposition Rate of species-n

(in the **Species...** category) is the amount of a surface species that is deposited on the substrate. Its unit quantity is **mass-flux**.

Surface dpdt RMS

(in the **Acoustics...** category) is the RMS value of the time-derivative of static pressure ($\partial p/\partial t$). It is available when the **Ffowcs-Williams & Hawkings** acoustics model is being used.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Heat Transfer Coef.

(in the **Wall Fluxes...** category), as defined in ANSYS FLUENT, is given by the equation

$$h_{eff} = \frac{q}{T_{wall} - T_{ref}} \tag{33–44}$$

T_{ref} is the reference temperature defined in the *Reference Values Task Page* (p. 1898). Please note that T_{ref} is a constant value that should be representative of the problem. Its unit quantity is the **heat-transfer-coefficient**.

Surface Incident Radiation

(in the **Wall Fluxes...** category) is the net incoming radiation heat flux on a surface. Its unit quantity is **heat-flux**.

Surface Nusselt Number

(in the **Wall Fluxes...** category) is a local nondimensional coefficient of heat transfer defined by the equation

$$Nu = \frac{h_{eff} L_{ref}}{k} \tag{33–45}$$

Surface Stanton Number

(in the **Wall Fluxes...** category) is a nondimensional coefficient of heat transfer defined by the equation

$$St = \frac{h_{eff}}{\rho_{ref} v_{ref} c_p} \tag{33–46}$$

v_{ref} are reference values of density and velocity defined in the *Reference Values Task Page* (p. 1898), and c_p is the specific heat at constant pressure.

Swirl Pull Velocity

(in the **Solidification/Melting...** category) is the tangential-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Swirl Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity in an axisymmetric swirling flow. See *Velocity Reporting Options* (p. 1531) for details. The unit quantity for **Swirl Velocity** is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Swirl-Wall Shear Stress

(in the **Wall Fluxes...** category) is the swirl component of the force acting tangential to the surface due to friction. Its unit quantity is **pressure**.

Tangential Velocity

(in the **Velocity...** category) is the velocity component in the tangential direction. (See *Velocity Reporting Options* (p. 1531) for details.) The unit quantity for **Tangential Velocity** is **velocity**.

Temperature...

indicates the quantities associated with the thermodynamic temperature of a material.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Conductivity

(in the **Properties...** category) is a parameter (k) that defines the conduction rate through a material via Fourier's law ($q = -k \nabla T$). A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator). Its unit quantity is **thermal-conductivity**.

Thermal Diff Coef of species-n

(in the **Species...** category) is the thermal diffusion coefficient for the nth species ($D_{T,i}$ in *Equation 8–58* (p. 439), *Equation 8–60* (p. 439), and *Equation 8–64* (p. 441)). Its unit quantity is **viscosity**.

Time Step

(in the **Residuals...** category) is the local time step of the cell, Δt, at the current iteration level. Its unit quantity is **time**.

Time Step Scale

(in the **Species...** category) is the factor by which the time step is reduced for the stiff chemistry solver (available in the density-based solver only). The time step is scaled down based on an eigenvalue and positivity analysis.

Total Energy

(in the **Temperature...** category) is the total energy per unit mass. Its unit quantity is **specific-energy**. For all species models, plots of **Total Energy** include the sensible, chemical and kinetic energies. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Enthalpy

(in the **Temperature...** category) is defined as $H + \frac{1}{2}v^2$ where H is the **Enthalpy**, as defined in Equation 5–7 in the Theory Guide, and v is the velocity magnitude. Its unit quantity is **specific-energy**. For all species models, plots of **Total Enthalpy** consist of the sensible, chemical and kinetic energies. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Enthalpy Deviation

(in the **Temperature...** category) is the difference between **Total Enthalpy** and the reference enthalpy, $H + \frac{1}{2}v^2 - H_{ref}$, where H_{ref} is the reference enthalpy defined in the *Reference Values Task Page* (p. 1898). However, for non-premixed and partially premixed models, **Total Enthalpy Deviation** is the difference between **Total Enthalpy** and total adiabatic enthalpy (total enthalpy where no heat loss or gain occurs). The unit quantity for **Total Enthalpy Deviation** is **specific-energy**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Pressure

(in the **Pressure...** category) is the pressure at the thermodynamic state that would exist if the fluid were brought to zero velocity and zero potential. For compressible flows, the total pressure is computed using isentropic relationships. For constant c_p, this reduces to:

$$p_0 = p\left[1 + \frac{\gamma - 1}{2}M^2\right]^{\gamma/(\gamma-1)} \tag{33–47}$$

For incompressible flows (constant density fluid), we use Bernoulli's equation, $p_0 = p + p_{dyn}$, where p_{dyn} is the local dynamic pressure. Its unit quantity is **pressure**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that in the postprocessing, the total pressure is presented as gauge pressure, for compressible and incompressible flows. If the total absolute pressure is needed, then add the value of the reference pressure to the total gauge pressure.

Total Surface Heat Flux

(in the **Wall Fluxes...** category) is the rate of total heat transfer through the control surface. It is calculated by the solver according to the boundary conditions being applied at that surface. By definition, heat flux out of the domain is negative, and heat flux into the domain is positive. The unit quantity for **Total Surface Heat Flux** is **heat-flux**.

Total Temperature

(in the **Temperature...** category) is the temperature at the thermodynamic state that would exist if the fluid were brought to zero velocity. For compressible flows, the total temperature is computed from the total enthalpy using the current c_p method (specified in the *Create/Edit Materials Dialog Box* (p. 1736)). For incompressible flows, the total temperature is equal to the static temperature, unless kinetic energy is explicitly added. The unit quantity for **Total Temperature** is **temperature**.

Transmitted Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux transmitted by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Transmitted Visible Solar Flux, Transmitted IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux transmitted by a semi-transparent wall or porous jump boundary for a visible or infrared radiation.

Turbulence...

includes quantities related to turbulence. See *Modeling Turbulence* (p. 643) for information about the turbulence models available in ANSYS FLUENT.

Turbulence Intensity

(in the **Turbulence...** category) is the ratio of the magnitude of the RMS turbulent fluctuations to the reference velocity:

$$I = \frac{\sqrt{\frac{2}{3}k}}{v_{ref}} \tag{33–48}$$

where k is the turbulence kinetic energy and v_{ref} is the reference velocity specified in the *Reference Values Task Page* (p. 1898). The reference value specified should be the mean velocity magnitude for the flow. Note that turbulence intensity can be defined in different ways, so you may want to use a custom field function for its definition. See *Custom Field Functions* (p. 1579) for more information.

Turbulent Dissipation Rate (Epsilon)

(in the **Turbulence...** category) is the turbulent dissipation rate. Its unit quantity is **turbulent-energy-diss-rate**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Turbulent Flame Speed

(in the **Premixed Combustion...** category) is the turbulent flame speed computed by ANSYS FLUENT using Equation 9–9 in the Theory Guide. Its unit quantity is **velocity**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Kinetic Energy (k)

(in the **Turbulence...** category) is the turbulence kinetic energy per unit mass defined as

$$k = \frac{1}{2}\overline{u'_i u'_i} \tag{33–49}$$

Turbulent Rate of Reaction-n

(in the **Reactions...** category) is the rate of progress of the nth reaction computed by Equation 7–24 or Equation 7–25 (in the Theory Guide). For the "eddy-dissipation" model, the value is the same as the **Rate of Reaction-n**. For the "finite-rate" model, the value is zero.

Turbulent Reynolds Number (Re_y)

(in the **Turbulence...** category) is a nondimensional quantity defined as

$$\frac{\rho d \sqrt{k}}{\mu_{lam}} \tag{33–50}$$

where k is turbulence kinetic energy, d is the distance to the nearest wall, and μ_{lam} is the laminar viscosity.

Turbulent Viscosity

(in the **Turbulence...** category) is the turbulent viscosity of the fluid computed using the turbulence model. Its unit quantity is **viscosity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Turbulent Viscosity Ratio

(in the **Turbulence...** category) is the ratio of turbulent viscosity to the laminar viscosity.

udm-n

(in the **User Defined Memory...** category) is the value of the quantity in the nth user-defined memory location.

Unburnt Fuel Mass Fraction

(in the **Premixed Combustion...** category) is the mass fraction of unburnt fuel. This function is available only for non-adiabatic models.

Unsteady Statistics...

includes mean and root mean square (RMS) values of solution variables derived from transient flow calculations.

User Defined Memory...

includes quantities that have been allocated to a user-defined memory location. See the separate UDF Manual for details about user-defined memory.

User-Defined Scalars...

includes quantities related to user-defined scalars. See the separate UDF Manual for information about using user-defined scalars.

UU Reynolds Stress

(in the **Turbulence...** category) is the $\overline{u'^2}$ stress.

UV Reynolds Stress

(in the **Turbulence...** category) is the $\overline{u'v'}$ stress.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{u' w'}$ stress.

Variance of Species

(in the **NOx...** category) is the variance of the mass fraction of a selected species in the flow field. It is calculated from Equation 14–113 in the Theory Guide.

Variance of Species 1, Variance of Species 2

(in the **NOx...** category) are the variances of the mass fractions of the selected species in the flow field. They are each calculated from Equation 14–113 in the Theory Guide.

Variance of Temperature

(in the **NOx...** category) is the variance of the normalized temperature in the flow field. It is calculated from Equation 14–113 in the Theory Guide.

Velocity...

includes the quantities associated with the rate of change in position with time. The instantaneous velocity of a particle is defined as the first derivative of the position vector with respect to time, $d\vec{r}/dt$, termed the velocity vector, $\vec{v}$.

Velocity Angle

(in the **Velocity...** category) is defined as follows:

For a 2D model,

$$\tan^{-1}\left(\frac{\text{y-velocity-component}}{\text{x-velocity-component}}\right) \tag{33–51}$$

For a 2D or axisymmetric model,

$$\tan^{-1}\left(\frac{\text{radial-velocity-component}}{\text{axial-velocity-component}}\right) \tag{33–52}$$

For a 3D model,

$$\tan^{-1}\left(\frac{\text{tangential-velocity-component}}{\text{axial-velocity-component}}\right) \tag{33–53}$$

Its unit quantity is **angle**.

Velocity Magnitude

(in the **Velocity...** category) is the speed of the fluid. Its unit quantity is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Volume fraction

(in the **Phases...** category) is the volume fraction of the selected phase in the **Phase** drop-down list.

Vorticity Magnitude

(in the **Velocity...** category) is the magnitude of the vorticity vector. Vorticity is a measure of the rotation of a fluid element as it moves in the flow field, and is defined as the curl of the velocity vector:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\xi = \nabla \times \vec{V} \tag{33–54}$$

VV Reynolds Stress

(in the **Turbulence...** category) is the $\overline{v'^2}$ stress.

VW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{v'w'}$ stress.

Wall Fluxes...

includes quantities related to forces and heat transfer at wall surfaces.

Wall Func. Heat Tran. Coef.

is defined by the equation

$$h_{eff} = \frac{\rho C_p C_\mu^{1/4} k_p^{1/2}}{T^*} \tag{33–55}$$

where C_p is the specific heat, k_p is the turbulence kinetic energy at point P, and T^* is the dimensionless law-of-the-wall temperature defined in Equation 4–278 in the Theory Guide.

Important

Note that ANSYS FLUENT reports wall functions heat transfer coefficient (*Equation 33–55* (p. 1578)) as zero for adiabatic walls.

Wall Shear Stress

(in the **Wall Fluxes...** category) is the force acting tangential to the surface due to friction. Its unit quantity is **pressure**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Wall Temperature (Inner Surface)

(in the **Temperature...** category) is the temperature on the inner surface of a wall (corresponding to the side of the wall surface away from the adjacent fluid or solid cell zone). Note that wall thermal boundary conditions are applied on this surface. See also *Figure 7.32* (p. 312). The unit quantity for **Wall Temperature (Inner Surface)** is **temperature**.

Wall Temperature (Outer Surface)

(in the **Temperature...** category) is the temperature on the outer surface of a wall (corresponding to the side of the wall surface toward the adjacent fluid or solid cell zone). Note that wall thermal boundary conditions are applied on the **Inner Surface**. See also *Figure 7.32* (p. 312). The unit quantity for **Wall Temperature (Outer Surface)** is **temperature**.

Wall Yplus

(in the **Turbulence...** category) is a nondimensional parameter defined by the equation

$$y^+ = \frac{\rho u_\tau y_P}{\mu} \tag{33–56}$$

where $u_\tau = \sqrt{\tau_w / \rho_w}$ is the friction velocity, y_P is the distance from point P to the wall, ρ is the fluid density, and μ is the fluid viscosity at point P. See Near-Wall Treatments for Wall-Bounded Turbulent

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flows in the Theory Guide for details. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Wall Ystar

(in the **Turbulence...** category) is a nondimensional parameter defined by the equation

$$y^* = \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu} \tag{33–57}$$

where k_P is the turbulence kinetic energy at point P, y_P is the distance from point P to the wall, ρ is the fluid density, and μ is the fluid viscosity at point P. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for details.

WW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{w'^2}$ stress.

X-Coordinate, Y-Coordinate, Z-Coordinate

(in the **Mesh...** category) are the Cartesian coordinates in the x-axis, y-axis, and z-axis directions respectively. The unit quantity for these variables is **length**.

X Face Area, Y Face Area, Z Face Area

(in the **Mesh...** category) are the components of the face area vector for noninternal faces (i.e., faces that only have `c0` and no `c1`). The values are stored on the face itself and used when required. These variables are intended only for zone surfaces and not for other surfaces created for postprocessing.

X Pull Velocity, Y Pull Velocity, Z Pull Velocity

(in the **Solidification/Melting...** category) are the x, y, and z components of the pull velocity for the solid material in a continuous casting process. The unit quantity for each is **velocity**.

X Velocity, Y Velocity, Z Velocity

(in the **Velocity...** category) are the components of the velocity vector in the x-axis, y-axis, and z-axis directions, respectively. The unit quantity for these variables is **velocity**. For multiphase models, these values correspond to the selected phase in the **Phase** drop-down list.

X-Vorticity, Y-Vorticity, Z-Vorticity

(in the **Velocity...** category) are the x, y, and z components of the vorticity vector.

X-Wall Shear Stress, Y-Wall Shear Stress, Z-Wall Shear Stress

(in the **Wall Fluxes...** category) are the x, y, and z components of the force acting tangential to the surface due to friction. The unit quantity for these variables is **pressure**. For multiphase models, these values correspond to the selected phase in the **Phase** drop-down list.

33.5. Custom Field Functions

In addition to the basic field variables provided by ANSYS FLUENT (and described in *Alphabetical Listing of Field Variables and Their Definitions* (p. 1547)), you can also define your own field functions to be used in conjunction with any of the commands that use these variables (contour and vector display, XY plots, etc.). This capability is available with the *Custom Field Function Calculator Dialog Box* (p. 2106). You can use the default field variables, previously defined calculator functions, and calculator operators to create new functions. (Several sample functions are described in *Sample Custom Field Functions* (p. 1583).)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Any field functions that you define will be saved in the case file the next time that you save it. You can also save your custom field functions to a separate file (as described in *Manipulating, Saving, and Loading Custom Field Functions* (p. 1582)), so that they can be used with a different case file.

Important

Note that all custom field functions are evaluated and stored in SI units.

Any solver-defined flow variables that you use in your field-function definition will be automatically converted if they are not already in SI units, but you must be careful to enter constants in the appropriate units. Note also that explicit node values are not available for custom field functions; all node values for these functions will be computed by averaging the values in the surrounding cells, as described in *Node Values* (p. 1529).

Important

When using the parallel version of ANSYS FLUENT, the only packages to which you can export custom field functions are the following:

- ANSYS CFD-Post
- EnSight Case Gold
- Fieldview Unstructured

For further information about exporting files, see *Exporting Solution Data* (p. 84).

For additional information, please see the following sections:

33.5.1. Creating a Custom Field Function
33.5.2. Manipulating, Saving, and Loading Custom Field Functions
33.5.3. Sample Custom Field Functions

33.5.1. Creating a Custom Field Function

To create your own field function, you will use the *Custom Field Function Calculator Dialog Box* (p. 2106) (*Figure 33.3* (p. 1581)). This dialog box allows you to define field functions based on existing functions, using simple calculator operators. Any functions that you define will be added to the list of default flow variables and other field functions provided by the solver.

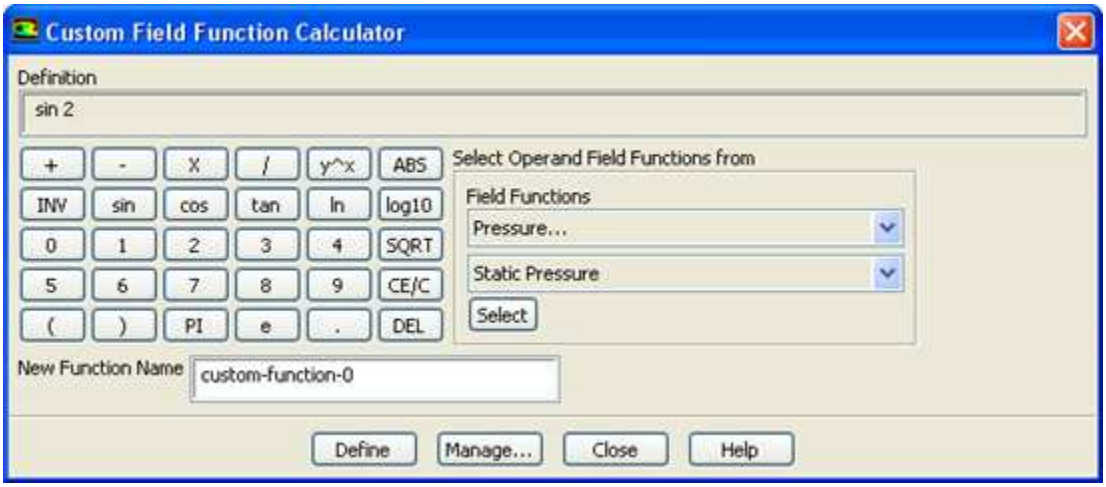
Define → Custom Field Functions...

Important

Recall that you must enter all constants in the function definition in SI units.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.3 The Custom Field Function Calculator Dialog Box



The steps for creating a custom field function are as follows:

1. Use the calculator buttons and the **Field Functions** list and **Select** button to specify the function definition, as described below. (As you select each item from the **Field Functions** list or click a button in the calculator keypad, its symbol will appear in the **Definition** text entry box. You *cannot* edit the contents of this box directly; if you want to delete part of a function, use the **DEL** button on the keypad.)

   Important

   The range of integers and real numbers that can be stored is as follows:

   $$-2147483648 > integers < 2147483647$$

   $$-1.79769e+308 > real < 1.79769e+308$$

   Note that using a number less than 1e-39 may produce inaccurate results, while values less than 1e-45 will produce a result of zero.

2. Specify the name of the function in the **New Function Name** field.

   Important

   Be sure that you do not specify a name that is already used for a standard field function (e.g., `velocity-magnitude`); you can see a complete list of the predefined field functions in ANSYS FLUENT by selecting the `display/contours` text command and viewing the available choices for `contours of`.

3. Click the **Define** button.

When you click **Define**, the solver will create the function and add it to the list of **Custom Field Functions** within the drop-down list of available field functions. The **Define** push button is grayed out after you create a new function or if the **Definition** text entry box is empty.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Should you decide to rename or delete the function after you have completed the definition, you can do so in the *Field Function Definitions Dialog Box* (p. 2107), which you can open by clicking on the **Manage...** push button. See *Manipulating, Saving, and Loading Custom Field Functions* (p. 1582) for details.

33.5.1.1. Using the Calculator Buttons

Your function definition can include many basic calculator operations (e.g., addition, subtraction, multiplication, square root). When you select a calculator button (by clicking on it), the appropriate symbol will appear in the **Definition** text entry box. The meaning of the buttons is straightforward; they are similar to the buttons you would find on any standard calculator. You should, however, note the following:

- The **CE/C** button will clear the entire **Definition** and the **New Function Name**, if you have entered one. The **DEL** button will delete only the last entry in the **Definition** text entry box. You can use **DEL** to delete characters one at a time, starting with the last one entered.
- To obtain the inverse trigonometric functions arcsin, arccos, and arctan, click the **INV** button before selecting **sin**, **cos**, or **tan**.
- The **ABS** button yields the absolute value of the number that follows it. Likewise, the **ln** button yields the natural logarithm of the number that follows it, and the **log10** button yields the base 10 logarithm function of the number that follows it.

  Important

  log10 and **ln** will be calculated for values greater than 0. For values less than or equal to 0, the resultant value will be zero.

- The **PI** button represents π and the **e** button represents the base of the natural logarithm system (which is approximately equal to 2.71828).

33.5.1.2. Using the Field Functions List

Your function definition can also include any of the field functions defined by the solver (and listed in *Alphabetical Listing of Field Variables and Their Definitions* (p. 1547)) or by you. To include one of these variables/functions in your function definition, select it in the **Field Functions** drop-down list and then click the **Select** button below the list. The symbol for the selected item will appear in the **Definition** text entry box (e.g., **p** will appear if you select **Static Pressure**).

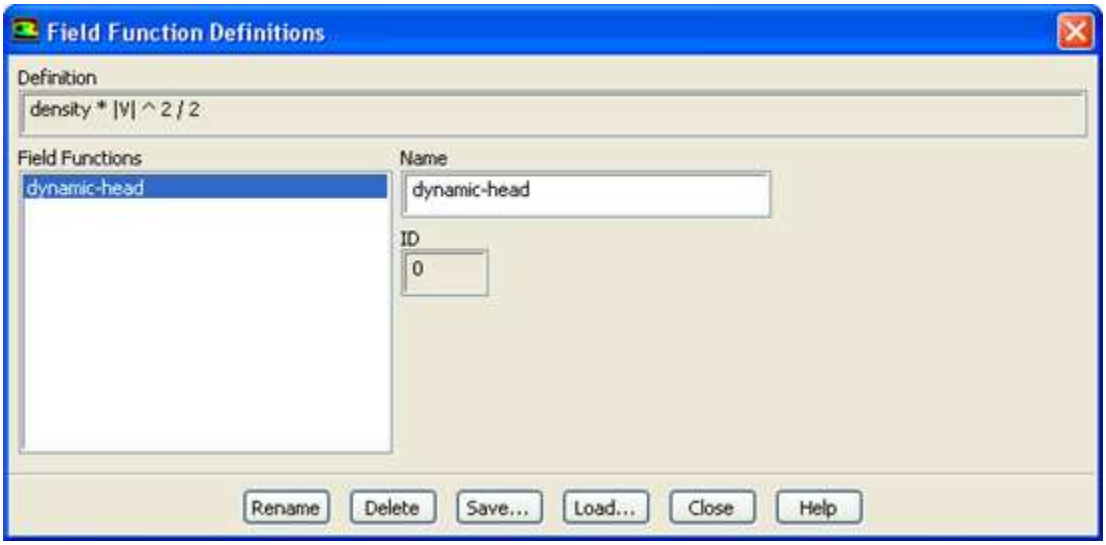
33.5.2. Manipulating, Saving, and Loading Custom Field Functions

Once you have defined your field functions, you can manipulate them using the *Field Function Definitions Dialog Box* (p. 2107) (*Figure 33.4* (p. 1583)). You can display a function definition to be sure that it is correct, delete the function if you decide that it is incorrect and needs to be redefined, or give the function a new name. You can also save custom field functions to a file or read them from a file. The custom field function file allows you to transfer your custom functions between case files.

To open the *Field Function Definitions Dialog Box* (p. 2107), click the **Manage...** button in the *Custom Field Function Calculator Dialog Box* (p. 2106).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.4 The Field Function Definitions Dialog Box



The following actions can be performed in the *Field Function Definitions Dialog Box* (p. 2107):

- To check the definition of a function, select it in the **Field Functions** list. Its definition will be displayed in the **Definition** field. This display is for informational purposes only; you cannot edit it. If you want to change a function definition, you must delete the function and define it again in the *Custom Field Function Calculator Dialog Box* (p. 2106).
- To delete a function, select it in the **Field Functions** list and click the **Delete** button.
- To rename a function, select it in the **Field Functions** list, enter a new name in the **Name** field, and click the **Rename** button.

Important

Be sure that you do not specify a name that is already used for a standard field function (e.g., `velocity-magnitude`); you can see a complete list of the predefined field functions in ANSYS FLUENT by selecting the `display/contours` text command and viewing the available choices for `contours of`.

- To save all of the functions in the **Field Functions** list to a file, click the **Save...** button and specify the file name in *The **Select File** Dialog Box* (p. 33).
- To read custom field functions from a file that you saved as described above, click the **Load...** button and specify the file name in the resulting **Select File** dialog box. (Custom field function files are valid Scheme functions, and can also be loaded with the **File/Read/Scheme...** menu item, as described in *Reading Scheme Source Files* (p. 73).)

33.5.3. Sample Custom Field Functions

When you are checking the results of your simulation, you may find it useful to define some of the following field functions:

- To define a function that determines the ratio of static pressure to inlet total pressure, use the relationship

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R = \frac{p + p_{op}}{p_{to} + p_{op}} \tag{33–58}$$

where p is the static pressure calculated by the solver, p_{to} is the inlet total pressure, and p_{op} is the operating pressure for the problem. Use the solver-defined function **Static Pressure** for p, and the numerical value that you specified for **Gauge Total Pressure** in the *Pressure Inlet Dialog Box* (p. 1848) for p_{to}. Specify the value of the operating pressure to be the value that you set in the *Operating Conditions Dialog Box* (p. 1806). As discussed in *Operating Pressure* (p. 449), all pressures in ANSYS FLUENT are gauge pressures relative to the operating pressure. If the operating pressure is zero, as is generally the case for compressible flow calculations, the expression for the pressure ratio reduces to

$$PR = \frac{p}{p_{to}} \tag{33–59}$$

- To define a function that determines the critical velocity ratio v/a_*, a parameter that is sometimes used in turbomachinery calculations, use the relationship

$$\frac{v}{a_*} = \left[\left(\frac{\gamma+1}{\gamma-1}\right)\left(1 - PR^{(\gamma-1)/\gamma}\right)\right]^{1/2} \tag{33–60}$$

  In this relationship, a_* is the critical velocity (i.e., the velocity that would occur for the same stagnation conditions if $M = 1$), γ is the ratio of specific heats, and PR is the pressure ratio defined in *Equation 33–59* (p. 1584) for which you created your own function. For γ, ratio of specific heats, select **Specific Heat Ratio (gamma)** in the **Properties...** category. To include PR, select **Custom Field Functions...** in the first drop-down list under **Field Functions**, and then select from the second list the function name that you assigned PR.

- Suppose you have swirling flow in a pipe, aligned with the z axis, and you want to calculate the flow rate of angular momentum through a cross-sectional plane:

$$\int \rho r v_\theta \vec{v} \cdot d\vec{A} \tag{33–61}$$

  You can create a function for the product $r v_\theta$, where r is the **Radial Coordinate** and v_θ is the **Tangential Velocity**. Then use the *Surface Integrals Dialog Box* (p. 2031) to compute the flow rate of this quantity.

Important

The custom field function containing model dependent functions (like temperature when the energy equation is enabled) will be computed only when those models are still active.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 34: Parallel Processing

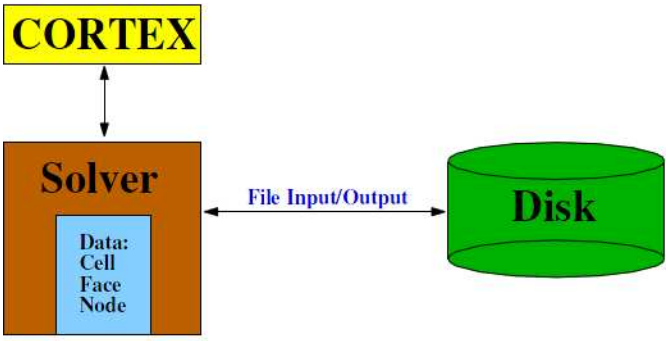
The following sections describe the parallel-processing features of ANSYS FLUENT.

34.1. Introduction to Parallel Processing
34.2. Starting Parallel ANSYS FLUENT Using FLUENT Launcher
34.3. Starting Parallel ANSYS FLUENT on a Windows System
34.4. Starting Parallel ANSYS FLUENT on a Linux/UNIX System
34.5. Mesh Partitioning and Load Balancing
34.6. Controlling the Threads
34.7. Checking Network Connectivity
34.8. Checking and Improving Parallel Performance

34.1. Introduction to Parallel Processing

The ANSYS FLUENT serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer (*Figure 34.1* (p. 1585)).

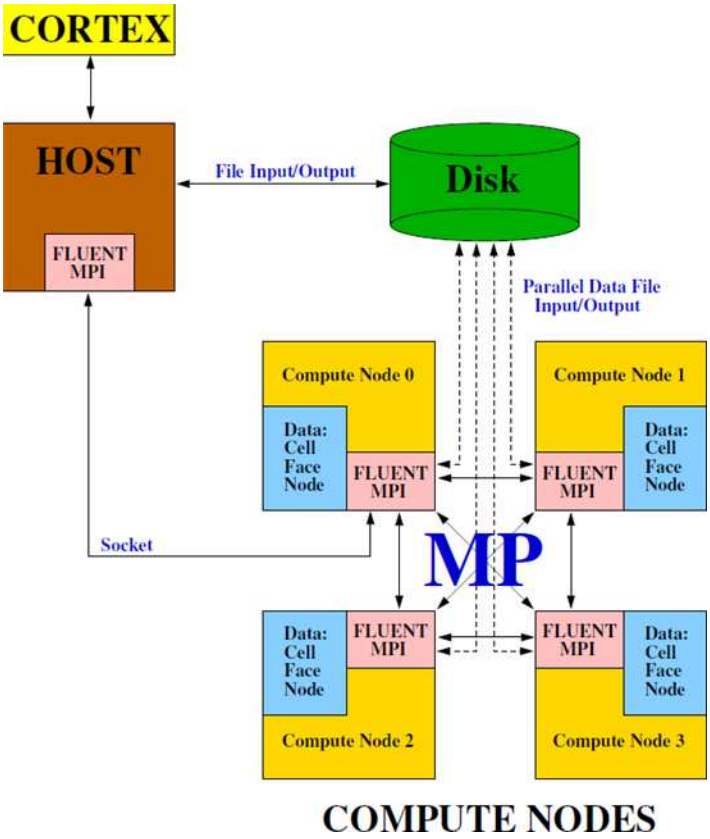
Figure 34.1 Serial ANSYS FLUENT Architecture



ANSYS FLUENT's parallel solver allows you to compute a solution by using multiple processes that may be executing on the same computer, or on different computers in a network (*Figure 34.2* (p. 1586)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.2 Parallel ANSYS FLUENT Architecture



Parallel processing in ANSYS FLUENT involves an interaction between ANSYS FLUENT, a host process, and a set of compute-node processes. ANSYS FLUENT interacts with the host process and the collection of compute nodes using a utility called `cortex` that manages ANSYS FLUENT's user interface and basic graphical functions.

Parallel ANSYS FLUENT splits up the mesh and data into multiple partitions, then assigns each mesh partition to a different compute process (or node). The number of partitions is an integral multiple of the number of compute nodes available to you (e.g., 8 partitions for 1, 2, 4, or 8 compute nodes). The compute-node processes can be executed on a massively-parallel computer, a multiple-CPU workstation, or a network cluster of computers.

Important

In general, as the number of compute nodes increases, turnaround time for the solution will decrease. However, parallel efficiency decreases as the ratio of communication to computation increases, so you should be careful to choose a large enough problem for the parallel machine.

ANSYS FLUENT uses a host process that does not contain any mesh data. Instead, the host process only interprets commands from ANSYS FLUENT's graphics-related interface, `cortex`.

The host distributes those commands to the other compute nodes via a socket interconnect to a single designated compute node called `compute-node-0`. This specialized compute node distributes the host commands to the other compute nodes. Each compute node simultaneously executes the same program on its own data set. Communication from the compute nodes to the host is possible only through `compute-node-0` and only when all compute nodes have synchronized with each other.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Each compute node is virtually connected to every other compute node, and relies on inter-process communication to perform such functions as sending and receiving arrays, synchronizing, and performing global operations (such as summations over all cells). Inter-process communication is managed by a message-passing library. For example, the message-passing library could be a vendor implementation of the Message Passing Interface (MPI) standard, as depicted in *Figure 34.2* (p. 1586).

All of the parallel ANSYS FLUENT processes (as well as the serial process) are identified by a unique integer ID. The host collects messages from `compute-node-0` and performs operations (such as printing, displaying messages, and writing to a file) on all of the data, in the same way as the serial solver. You have the option of bypassing the host when inputting or outputting parallel data files, so that the files are passed directly between the compute nodes and the disk in a parallel fashion. This can reduce the time for data file I/O operations. (For details, see *Reading and Writing Parallel Data Files* (p. 68)).

For additional information, please see the following section:
34.1.1. Recommended Usage of Parallel ANSYS FLUENT

34.1.1. Recommended Usage of Parallel ANSYS FLUENT

The recommended procedure for using parallel ANSYS FLUENT is as follows:

1. Start up the parallel solver. For details, see *Starting Parallel ANSYS FLUENT on a Windows System* (p. 1594) and *Starting Parallel ANSYS FLUENT on a Linux/UNIX System* (p. 1599).
2. Read your case file and have ANSYS FLUENT partition the mesh automatically upon loading it.
3. Review the partitions and perform partitioning again, if necessary. See *Checking the Partitions* (p. 1622) for details on checking your partitions. Note that there are other approaches for partitioning, including manual partitioning in either the serial or the parallel solver. For details, see *Mesh Partitioning and Load Balancing* (p. 1603).
4. Calculate a solution. See *Checking and Improving Parallel Performance* (p. 1626) for information on checking and improving the parallel performance.

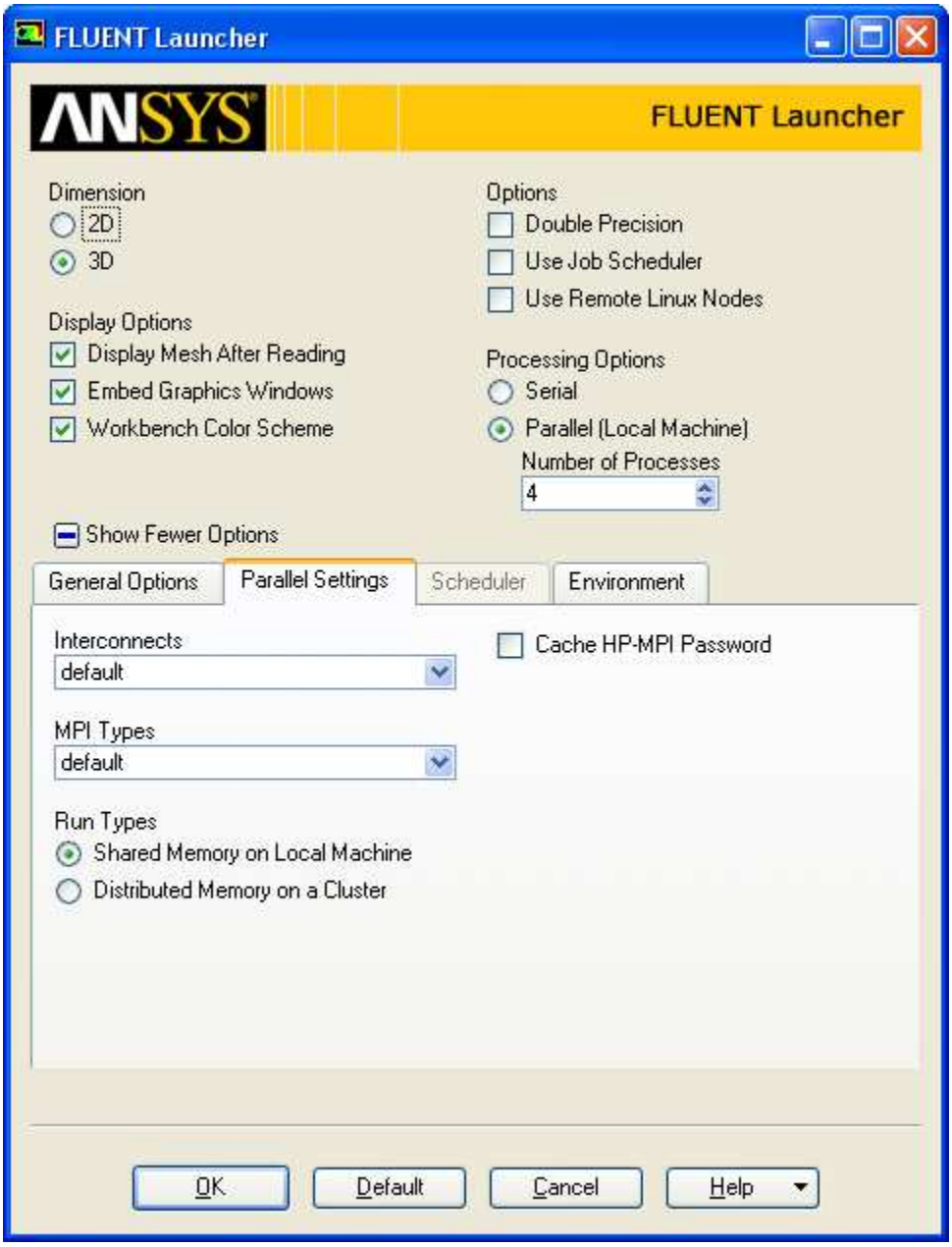
34.2. Starting Parallel ANSYS FLUENT Using FLUENT Launcher

Whether you start ANSYS FLUENT either from the Linux/UNIX or Windows command line with no arguments, from the Windows Programs menu, or from the Windows desktop, FLUENT Launcher will appear. (For details, see *Starting ANSYS FLUENT Using FLUENT Launcher* (p. 2)), where you can specify the dimensionality of the problem (2D or 3D), as well as other options (e.g., whether you want a single-precision or double-precision calculation).

Parallel calculation options can be set up by selecting **Parallel** under **Processing Options** in FLUENT Launcher. Once you select the **Parallel** option, you can also specify the number of processes using the **Number of Processes** field. Activating the **Parallel** option enables the **Parallel Settings** tab (visible when you select the **Show More Options** button). The **Parallel Settings** tab allows you to specify settings for running ANSYS FLUENT in parallel.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.3 The Parallel Settings Tab of FLUENT Launcher



- Specify the interconnect or system in the **Interconnects** drop-down list. The default setting is recommended. For a symmetric multi-processor (SMP) system, the default setting uses shared memory for communication.

  If you prefer to select a specific interconnect, you can choose either **ethernet**, **myrinet**, or **infiniband**. For more information about these interconnects, see *Table 34.1: Supported Interconnects for the Windows Platform* (p. 1596), *Table 34.2: Available MPIs for Windows Platforms* (p. 1596), and *Table 34.3: Supported MPIs for Windows Architectures (Per Interconnect)* (p. 1597).

- Specify the type of message passing interface (MPI) you require for the parallel computations in the **MPI Types** field. The list of MPI types varies depending on the selected release and the selected architecture. There are several options, based on the operating system of the parallel cluster. For more information about the available MPI types, see *Table 34.1: Supported Interconnects for the Windows Platform* (p. 1596) - *Table 34.2: Available MPIs for Windows Platforms* (p. 1596).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

It is your responsibility to make sure the interconnects and the MPI types are compatible. If incompatible inputs are used, FLUENT Launcher resorts to using the default values.

- (Linux/UNIX Only) Specify either **RSH** (remote shell client) or **SSH** (secure shell client) under **Remote Spawn Command**. For more information about setting up your remote shell clients and secure shell clients, see *Setting Up Your Remote Shell and Secure Shell Clients* (p. 1602).
- Specify the type of parallel calculation under **Run Types**:
  - Select **Shared Memory on Local Machine** if the parallel calculations are performed by sharing memory allocations on your local machine.
  - Select **Distributed Memory on a Cluster** if the parallel calculations will be distributed among several machines.

    You can select **Machine Names** and enter the machine names directly into the text field. Machine names can be separated either by a comma or a space. This is not recommended for a long list of machine names.

    Alternatively, you can select **File Containing Machine Names** to specify a hosts file (a file that contains the machine names), or you can use the button to browse for a hosts file.

    To edit an existing hosts file, click the button.
- Specify if you would like to cache the password for the HP-MPI or not.

  Select the **Cache HP-MPI Password** option if you would like to save the required password to use the HP MPI type.
- Select **Use Job Scheduler** under **Options** if the parallel calculations are to be performed using a designated Job Scheduler. (For details, see *Setting Parallel Scheduler Options in FLUENT Launcher* (p. 1589)). This also enables the **Scheduler** tab of FLUENT Launcher.

For additional information, please see the following sections:

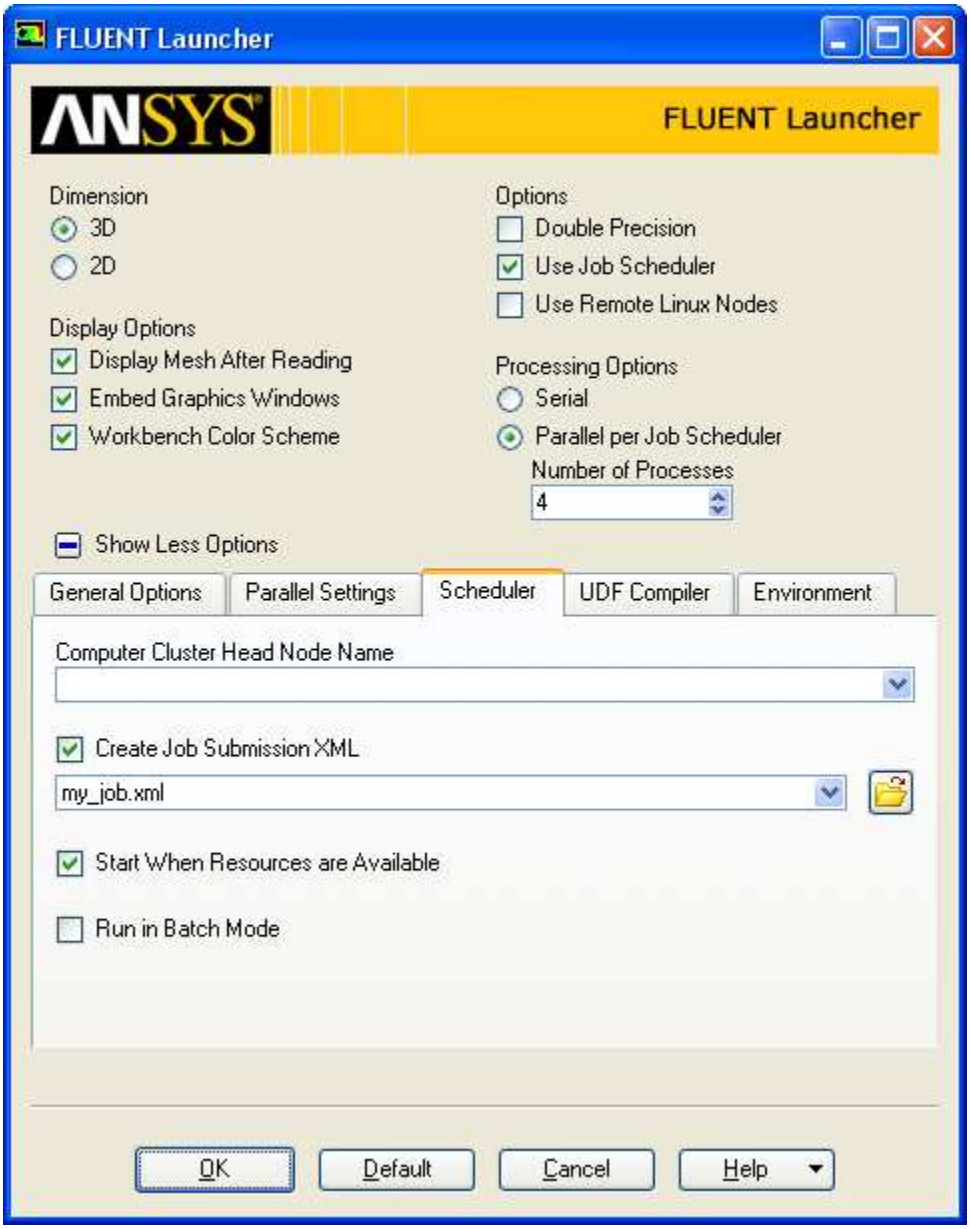
34.2.1. Setting Parallel Scheduler Options in FLUENT Launcher
34.2.2. Setting Additional Options When Running on Remote Linux Machines

34.2.1. Setting Parallel Scheduler Options in FLUENT Launcher

Activating the **Use Job Scheduler** option under **Options** in FLUENT Launcher enables the **Scheduler** tab (visible when you select **Show More Options**). The **Scheduler** tab allows you to specify settings for running ANSYS FLUENT with various job schedulers (e.g., the Microsoft Job Scheduler for Windows, or LSF, SGE, and PBS Pro on Linux/UNIX).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.4 The Scheduler Tab of FLUENT Launcher (Windows Version)



For the Windows 32-bit HP-MPI, Windows 64-bit CCP and HP-MPI, you can specify that you want to use the Microsoft Job Scheduler by selecting the **Use Job Scheduler** check box under **Options** in FLUENT Launcher. Once selected, you can then enter a machine name in the **Compute Cluster Head Node Name** text field in the **Scheduler** tab. If you are running ANSYS FLUENT on the head node, then you can keep the field empty. This option translates into the proper parallel command line syntax for using the Microsoft Job Scheduler (For details, see *Starting Parallel ANSYS FLUENT with the Microsoft Job Scheduler* (p. 1597)).

If you want ANSYS FLUENT to start after the necessary resources have been allocated by the Scheduler, then select the **Start When Resources are Available** check box.

To create the job submission XML file at the time when you launch ANSYS FLUENT, select the **Create Job Submission XML** option (available once you specify a journal file in the **General Options** tab) and enter the name of the XML file in the text field, or click to browse for an appropriate XML file. The XML file can be used to submit jobs to ANSYS FLUENT without having to use FLUENT Launcher.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Linux/UNIX, select the **Use Job Scheduler** check box under **Options** to use one of three available job schedulers in the **Scheduler** tab.

- Select the **Use LSF** radio button to use the LSF load management system with or without checkpointing. If you select **Use Checkpointing**, then you can specify a checkpointing directory in the **Checkpointing Directory** field. By default, the current working directory is used. In addition, you can specify a numerical value for the frequency of automatic checkpointing in the **Automatic Checkpoint with Setting of Period** field.

  For more information, go to the Documentation page (`http://fluentusers.com/fluent/doc/doc_f.htm`) on the User Services Center (`www.fluentusers.com`) and locate the Load Management Documentation page.
- Select the **Use SGE** radio button to use the SGE load management system. You can choose to set values for the **SGE qmaster**, as well as the **SGE queue**, or the **SGE pe**. Alternatively, you can select **Use SGE settings** check box and specify the location and name of the SGE configuration file.

  For more information, go to the Documentation page (`http://fluentusers.com/fluent/doc/doc_f.htm`) on the User Services Center (`www.fluentusers.com`) and locate the Load Management Documentation page.
- Select the **Use PBS Pro** radio button to use the PBS Pro load management system. For more information, go to the Documentation page (`http://fluentusers.com/fluent/doc/doc_f.htm`) on the User Services Center (`www.fluentusers.com`) and locate the Load Management Documentation page.

For Windows, you also have the ability to run in batch mode (using the **Run in Batch Mode** check box) when you provide a journal file (designated in the **General Options** tab) which exits ANSYS FLUENT at the end of the run.

For machines running the Windows HPC 2008 Server Scheduler, you also have the following options to choose from:

- **Job Template** allows you to create a custom submission policy to define the job parameters for an application. The cluster administrator can use job templates to manage job submission and optimize cluster usage.
- **Node Group** allows you to specify a collection of nodes. Cluster administrators can create groups and assign nodes to one or more groups.
- **Processor Unit** allows you to choose the following:
  - **Core** refers to a single, named host in the cluster.
  - **Socket** refers to a set of processors with a dedicated memory bus. This is also known as a non-uniform memory access (NUMA) node.
  - **Node** refers to an individual CPU on a node. For example, a dual-core processor is considered two cores.

For more information about running FLUENT jobs using the Windows HPC 2008 Server Scheduler, see the Frequently Asked Questions section of the User Services Center (`www.fluentusers.com`).

34.2.2. Setting Additional Options When Running on Remote Linux Machines

The **Remote** tab allows you to specify settings for running ANSYS FLUENT parallel simulations on Linux clusters, via the Windows interface.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.5 The Remote Tab of FLUENT Launcher

You can run simulations on Linux machines, either in serial or on parallel Linux clusters, via the Windows interface. To access remote 64-bit Linux clusters for your parallel calculation, select the **Parallel (Local Machine)** option under **Processing Options** (For details, see *Setting Parallel Options in FLUENT Launcher* (p. 5)), then enable **Use Remote Linux Nodes**, which appears under **Options**. The **Remote** tab in FLUENT Launcher will become available, where you can specify the remote ANSYS FLUENT Linux installation root path in the **Remote FLUENT Root Path** field. The **Remote Working Directory** field allows you to specify a working directory for the remote Linux nodes, other than the default `temp` directory.

Select one of the following **Remote Spawn Commands** to connect to the remote node:

- **RSH** (the default) is used to spawn nodes from the local Windows machine to the Linux head node as well as from the Linux head node to the compute nodes. If you want the Linux cluster to use SSH, then

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you need to set the ANSYS FLUENT_NO_REMOTE_RSH to 1. You will also need to set up passwordless access.

- **SSH** is used to spawn nodes from the local Windows machine to the Linux head node as well as from the Linux head node to the compute nodes. To use SSH with ANSYS FLUENT, you need to set up passwordless SSH access. If you want the Linux cluster to use RSH, then you need to set the FLUENT_NO_REMOTE_SSH to 1. For more information about setting up SSH without a password, see www.debian-administration.org/articles/152.
- **Other** allows you to provide other compatible remote shell commands.

Enable the **Use Remote Cluster Head Node** field and specify the remote node to which ANSYS FLUENT will connect for spawning (e.g., via rsh or ssh). If this is not provided, then ANSYS FLUENT will try to use the first machine in the machine file. If SGE is chosen as the job scheduler, then the **SGE qmaster** will serve the same purpose.

In addition to using the settings in the **Remote** tab in FLUENT Launcher, the following command line options are also available when starting ANSYS FLUENT from the command line:

-nodepath=path
is the path on the remote machine where ANSYS FLUENT is installed.

-node0=machine name
is the machine from which to launch other nodes.

-nodehomedir=directory
is the directory that becomes the current working directory for all the nodes. Additionally, this will be used as a scratch area for temporary files that are created on the nodes.

-rsh=remote shell command
is the command that will be used to launch executables remotely. This option defaults to rsh.exe but can point to any equivalent program. The form of this command should be that it should not wait for additional inputs such as passwords. For example, if you install SSH, and try to launch in mixed mode using ssh, the launch may fail unless you have set up a login for SSH without a password. For more information about setting up SSH without a password, see www.debian-administration.org/articles/152.

As there are known issues with launching ANSYS FLUENT in mixed Windows/Linux mode from cygwin, it is recommended that you use the command prompt (cmd.exe).

While ANSYS FLUENT case and data files are read from and written to the Windows machine while running mixed Windows/Linux simulations, parallel data files (.pdat files) are written by the nodes and will not be available on the Windows machine. Therefore, the path specified for writing .pdat files should be a Linux path (you can set node's home directory if you want to provide relative paths). If you use the graphical user interface to read and/or write .pdat files, the directory component of the path is ignored and only the file name is used to read and/or write the .pdat file relative to the node's current working directory.

When working with mixed Linux and Windows runs that employ user-defined functions (UDFs), you should keep in mind that the file that you have opened for reading/writing on the host machine will not be available on remote nodes and vice-versa. You may therefore have to transfer data present on the nodes to the host and write it from host, (or distribute the data from the host to the nodes after reading the data from the host).

34.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines

By selecting the **Use Remote Linux Nodes** option and the **Use Job Scheduler** option in FLUENT Launcher, you can set job scheduler options for the remote Linux machines you are accessing for your CFD analysis.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When these options are enabled in FLUENT Launcher, you can use the **Scheduler** tab to set parameters for either **LSF**, **SGE**, or **PBS Pro** job schedulers. You can learn more about each of the schedulers by referring to the Load Management Documentation.

The following list describes the various controls that are available in the **Scheduler** tab:

Use LSF
allows you to use the LSF job scheduler.

LSF queue
allows you to specify a job queue and enter the queue name in the text box.

Use Checkpointing
allows you to use checkpointing with LSF. By default, the checkpointing directory will be the current working directory; however, you have the option of enabling **Checkpointing Directory**.

Checkpointing Directory
allows you to specify a checkpointing directory which is different from the current working directory.

Automatic Checkpoint with Setting of Period
allows you to specify that the checkpointing is done automatically at a set time interval. Enter the period (in minutes) in the text box, otherwise checkpointing will not occur unless you call the `bchkpnt` command.

Use SGE
allows you to use the SGE job scheduler.

SGE qmaster
is the machine in the SGE job submission host list. SGE will allow the **SGE qmaster** node to summon jobs. By default, **localhost** is specified for **SGE qmaster**. Note that the button allows you to check the job status.

SGE queue
is the queue where you want to submit your ANSYS FLUENT jobs. Note that you can use the button to contact the **SGE qmaster** for a list of queues. Leave this field blank if you want to use the default queue.

SGE pe
is the parallel environment where you want to submit your ANSYS FLUENT jobs. The parallel environment must be defined by an administrator. For more information about creating a parallel environment, please refer to the SGE documentation. Leave this field blank if you want to use the default parallel environment.

Use PBS Pro
allows you to use the PBS Pro job scheduler.

34.3. Starting Parallel ANSYS FLUENT on a Windows System

You can run ANSYS FLUENT on a Windows system using either the graphical user interface (For details, see *Starting Parallel ANSYS FLUENT Using FLUENT Launcher* (p. 1587)) or command line options (For details, see *Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options* (p. 1595)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

See the separate installation instructions for more information about installing parallel ANSYS FLUENT for Windows. The startup instructions below assume that you have properly set up the necessary software, based on the appropriate installation instructions.

Additional information about installation issues can also be found in the Frequently Asked Questions section of the User Services Center (`www.fluentusers.com`).

For additional information, please see the following section:
34.3.1. Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options

34.3.1. Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options

To start the parallel version of ANSYS FLUENT using command line options, you can use the following syntax in a Command Prompt window:

`fluent` *version* `-t`*nprocs* `[-p`*interconnect* `] [-mpi=`*mpi_type* `] -cnf=` *hosts_file*

where

- *version* must be replaced by the version of ANSYS FLUENT you want to run (`2d`, `3d`, `2ddp`, or `3ddp`).
- `-p`*interconnect* (optional) specifies the type of interconnect. The `ethernet` interconnect is used by default if the option is not explicitly specified. See *Table 34.1: Supported Interconnects for the Windows Platform* (p. 1596), *Table 34.2: Available MPIs for Windows Platforms* (p. 1596), and *Table 34.3: Supported MPIs for Windows Architectures (Per Interconnect)* (p. 1597) for more information.
- `-mpi=`*mpi_type* (optional) specifies the MPI implementation. If the option is not specified, the default MPI for the given interconnect (HP MPI) will be used (the use of the default MPI is recommended). The available MPIs for Windows are shown in *Table 34.2: Available MPIs for Windows Platforms* (p. 1596).
- `-cnf=`*hosts_file* specifies the hosts file, which contains a list of the computers on which you want to run the parallel job. If the hosts file is not located in the folder where you are typing the startup command, you will need to supply the full pathname to the file.

  You can use a plain text editor such as Notepad to create the hosts file. The only restriction on the filename is that there should be no spaces in it. For example, `hosts.txt` is an acceptable hosts file name, but `my hosts.txt` is not.

  Your hosts file (e.g., `hosts.txt`) might contain the following entries:

  ```
  computer1
  computer2
  ```

  Important

  The last entry must be followed by a blank line.

  If a computer in the network is a multiprocessor, you can list it more than once. For example, if `computer1` has 2 CPUs, then, to take advantage of both CPUs, the `hosts.txt` file should list `computer1` twice:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
computer1
computer1
computer2
```

- `-t`*nprocs* specifies the number of processes to use. When the `-cnf` option is present, the *hosts_file* argument is used to determine which computers to use for the parallel job. For example, if there are 8 computers listed in the hosts file and you want to run a job with 4 processes, set *nprocs* to 4 (i.e., `-t4`) and ANSYS FLUENT will use the first 4 machines listed in the hosts file. Note that this does not apply to the Compute Cluster Server (CCS).

For example, the full command line to start a 3D parallel job on the first 4 computers listed in a hosts file called `hosts.txt` is as follows:

```
fluent 3d -t4 -cnf=hosts.txt
```

As another example, the full command line to start a 3D symmetrical multiprocessing (SMP) parallel job on 4 computers is as follows:

```
fluent 3d -t4
```

In either case, the default interconnect (`ethernet`) and the default communication library (`HP-MPI`) will be used since these options are not specified.

The first time that you try to run ANSYS FLUENT in parallel, you will be prompted for information about the current Windows account. For more information, see the HP-MPI setup FAQ in the Frequently Asked Questions section of the User Services Center (`www.fluentusers.com`).

The supported interconnects for dedicated parallel `ntx86` and `win64` Windows machines, the associated MPIs for them, and the corresponding syntax are listed in *Table 34.1: Supported Interconnects for the Windows Platform* (p. 1596) - *Table 34.3: Supported MPIs for Windows Architectures (Per Interconnect)* (p. 1597).

Table 34.1 Supported Interconnects for the Windows Platform

Platform	Processor	Architecture	Interconnects
Windows	32-bit	`ntx86`	`ethernet` (default)
	64-bit	`win64`	`ethernet` (default), `infiniband`, `myrinet`

Table 34.2 Available MPIs for Windows Platforms

MPI	Syntax (flag)	Communication Library	Notes
`hp`	`-mpi=hp`	HP-MPI	(1), (2)
`ms`	`-mpi=ms`	Microsoft MPI	(1), (2)
`intel`	`-mpi=intel`	Intel MPI	(1), (2)

(1) Used with Shared Memory Machine (SHM) where the memory is shared between the processors on a single machine.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(2) Used with Distributed Memory Machine (DMM) where each processor has it's own memory associated with it.

Table 34.3 Supported MPIs for Windows Architectures (Per Interconnect)

Architecture	Ethernet	Myrinet	Infiniband
`ntx86`	`hp` (default), `intel`	-	-
`win64`	`hp` (default), `intel`, `ms`	`ms`	`ms`

34.3.1.1. Starting Parallel ANSYS FLUENT with the Microsoft Job Scheduler

The Microsoft Job Scheduler allows you to manage multiple jobs and tasks, allocate computer resources, send tasks to compute nodes, and monitor jobs, tasks, and compute nodes.

ANSYS FLUENT currently supports Windows XP, Vista, as well as the Windows Server operating systems. The Windows Server operating systems include a compute cluster server (CCS) and a high performance computing server (HPC) that combines the Microsoft MPI type (`msmpi`) with the Microsoft Job Scheduler. ANSYS FLUENT provides a means of using the Microsoft Job Scheduler using the following flag in the parallel command:

`-ccp` *head-node-name*

where `-ccp` indicates the use of the compute cluster server package, and *head-node-name* indicates the name of the head node of the computer cluster.

For example, if you want to use the Microsoft Job Scheduler to run a 3D model on 2 nodes, the corresponding command syntax would be:

`fluent 3d -t2 -ccp` *head-node-name*

Important

Both the HP-MPI type (`hpmpi`) and the Intel MPI (`intel`) are not supported with the Microsoft Job Scheduler.

Note

When using Microsoft Job Scheduler, the best interconnect is automatically selected by MS-MPI and the default Ethernet option does not apply.

Though the usage described previously is recommended as an initial starting point for running ANSYS FLUENT with the Microsoft Job Scheduler, there are further options provided to meet your specific needs. ANSYS FLUENT allows you to do any of the following with the Microsoft Job Scheduler:

- Request resources from the Microsoft Job Scheduler first, before you launch ANSYS FLUENT.

  This is done by first submitting a job that will run until canceled, as shown in the following example:

  `job new/scheduler:`*head-node-name* `/numprocessors:2 /rununtilcanceled:true`

  This example requests a 2-node resource on a cluster named *head-node-name*. You will see that a job is created with the job ID *job-id*:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`job submit/scheduler:` *head-node-name* `/id:` *job-id*

Then check if the resources have been allocated:

`job view` *job-id* `/scheduler:` *head-node-name*

If the resources are ready, you can start ANSYS FLUENT using the job ID:

`fluent 3d -t2 -ccp` *head-node-name* `-jobid=`*job-id*

This job will be reusable until you decide to cancel it, at which point you must enter the following:

`job cancel` *job-id* `/scheduler:` *head-node-name*

- Have ANSYS FLUENT submit a CCS job, but delay the launching of ANSYS FLUENT until the actual resources are allocated.

  This is done by specifying the job ID as `-1`, as shown in the following example:

  `fluent 3d -t2 -ccp` *head-node-name* `-jobid=-1`

  If you want to stop the job application, click the **Cancel** button. ANSYS FLUENT will prompt you for confirmation, and then clean up the pending job and exit.

- Run your job using XML template files.

  This is done by first creating an XML template file, such as shown in the following example:

```
<?xml version="1.0" encoding="utf-8"?>
<Job
xmlns:m:xsi="http://www.w3.org/2001/XMLSchema-instance"
xmlns:m:xsd="http://www.w3.org/2001/XMLSchema"
SoftwareLicense=""
MaximumNumberOfProcessors="4"
MinimumNumberOfProcessors="4"
Runtime="Infinite"
IsExclusive="true"
Priority="Normal"
Name="Name_of_job"
Project="Fluent runs"
RunUntilCanceled="false">
<Tasks xmlns:m="http://www.microsoft.com/ComputeCluster/">
 <Task
   MaximumNumberOfProcessors="4"
   MinimumNumberOfProcessors="4"
   Depend=""
   WorkDirectory="\\file-server\home\user"
   Stdout="fluent-case.%CCP_JOBID%.out"
   Stderr="fluent-case.%CCP_JOBID%.err"
   Name="My Task"
   CommandLine="\\head-node\fluent-sharename\win64\fluent.exe 3d -i bsi.jou -t4"
   IsExclusive="true"
   IsRerunnable="false"
   Runtime="Infinite">
 </Task>
</Tasks>
</Job>
```

  where `fluent-sharename` is the name of the shared directory pointing to where ANSYS FLUENT is installed (e.g., `C:\Program Files\ANSYS Inc\v130\fluent`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that you must create a journal file that exits ANSYS FLUENT at the end of the run, and refer to it using the `-i` flag in your XML template file (`bs1.jou` in the previous example).

After you have saved the file and given it a name (e.g., `job1.xml`), you can submit the job as shown:

```
job submit /jobfile:job1.xml
```

- Run the job in batch mode without displaying the ANSYS FLUENT GUI.

  The following is an example of such a batch mode job:

  `job submit /scheduler:`*head-node-name*

  `/numprocessors:2 /workdir:\\file-server\home\user\`

  `\\head-node\fluent-sharename\ntbin\win64\fluent.exe 3d -t2 -i bs1.jou`

  where `fluent-sharename` is the name of the shared directory pointing to where ANSYS FLUENT is installed (e.g., `C:\Program Files\ANSYS Inc\v130\fluent`).

  Important

  – Note that you must create a journal file that exits ANSYS FLUENT at the end of the run, and refer to it using the `-i` flag in your batch mode job submission (`bs1.jou` in the previous example).

  – You can start ANSYS FLUENT jobs from any machine on which is installed either the full CCP or the CCP client tools, but note that all the machines must have the same version installed.

34.4. Starting Parallel ANSYS FLUENT on a Linux/UNIX System

You can run ANSYS FLUENT on a Linux/UNIX system using either the graphical user interface (For details, see *Starting Parallel ANSYS FLUENT Using FLUENT Launcher* (p. 1587)) or command line options (For details, see *Starting Parallel ANSYS FLUENT on a Linux/UNIX System Using Command Line Options* (p. 1599) *Setting Up Your Remote Shell and Secure Shell Clients* (p. 1602)).

For additional information, please see the following sections:

34.4.1. Starting Parallel ANSYS FLUENT on a Linux/UNIX System Using Command Line Options
34.4.2. Setting Up Your Remote Shell and Secure Shell Clients

34.4.1. Starting Parallel ANSYS FLUENT on a Linux/UNIX System Using Command Line Options

To start the parallel version of ANSYS FLUENT using command line options, you can use the following syntax in a command prompt window:

`fluent` *version* `-t`*nprocs* `[-p`*interconnect* `] [-mpi=`*mpi_type* `] -cnf=` *hosts_file*

where

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *version* must be replaced by the version of ANSYS FLUENT you want to run (`2d`, `3d`, `2ddp`, or `3ddp`).
- `-p`*interconnect* (optional) specifies the type of interconnect. The `ethernet` interconnect is used by default if the option is not explicitly specified. See *Table 34.4: Supported Interconnects for Linux/UNIX Platforms (Per Platform)* (p. 1601), *Table 34.5: Available MPIs for Linux/UNIX Platforms* (p. 1601), and *Table 34.6: Supported MPIs for Linux/UNIX Architectures (Per Interconnect)* (p. 1601) for more information.
- `-mpi=`*mpi_type* (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect will be used (the use of the default MPI is recommended). The available MPIs for Linux/UNIX are shown in *Table 34.5: Available MPIs for Linux/UNIX Platforms* (p. 1601).
- `-cnf=`*hosts_file* specifies the hosts file, which contains a list of the computers on which you want to run the parallel job. If the hosts file is not located in the directory where you are typing the startup command, you will need to supply the full pathname to the file.

  You can use a plain text editor to create the hosts file. The only restriction on the filename is that there should be no spaces in it. For example, `hosts.txt` is an acceptable hosts file name, but `my hosts.txt` is not.

  Your hosts file (e.g., `hosts.txt`) might contain the following entries:

  ```
  computer1
  computer2
  ```

  Important

  The last entry must be followed by a blank line.

  If a computer in the network is a multiprocessor, you can list it more than once. For example, if `computer1` has 2 CPUs, then, to take advantage of both CPUs, the `hosts.txt` file should list `computer1` twice:

  ```
  computer1
  computer1
  computer2
  ```

- `-t`*nprocs* specifies the number of processes to use. When the `-cnf` option is present, the *hosts_file* argument is used to determine which computers to use for the parallel job. For example, if there are 10 computers listed in the hosts file and you want to run a job with 5 processes, set *nprocs* to 5 (i.e., `-t5`) and ANSYS FLUENT will use the first 5 machines listed in the hosts file.

For example, to use the Infiniband interconnect, and to start the 3D solver with 4 compute nodes on the machines defined in the text file called `fluent.hosts`, you can enter the following in the command prompt:

```
fluent 3d -t4 -pinfiniband -cnf=fluent.hosts
```

Note that if the optional `-cnf=` *hosts_file* is specified, a compute node will be spawned on each machine listed in the file *hosts_file*. (If you enter this optional argument, do not include the square brackets.)

Also, ANSYS FLUENT provides a fault-tolerance feature on Infiniband Linux clusters running OFED. To invoke this feature, use the command line flag `-pinfiniband.ofedft` (or `-pib.ofedft`) which enables transparent port fail-over and high-availability features using HP-MPI. Note that while the simulations proceed more robustly with this option, there may be some degradation in performance.

The supported interconnects for parallel Linux/UNIX machines are listed below (*Table 34.4: Supported Interconnects for Linux/UNIX Platforms (Per Platform)* (p. 1601), *Table 34.5: Available MPIs for Linux/UNIX Platforms* (p. 1601),

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and *Table 34.6: Supported MPIs for Linux/UNIX Architectures (Per Interconnect)* (p. 1601)), along with their associated communication libraries, the corresponding syntax,

Table 34.4 Supported Interconnects for Linux/UNIX Platforms (Per Platform)

Platform	Processor	Architecture	Interconnects/Systems*
Linux	32-bit	`lnx86`	`ethernet`, `infiniband`, `myrinet`
	64-bit	`lnamd64`	`ethernet`, `infiniband`, `myrinet`
	64-bit Itanium	`lnia64`	`ethernet`, `infiniband`, `myrinet`, `altix`
Sun	64-bit	`ultra_64`	`vendor`** (default), `ethernet`
HP	64-bit Itanium	`hpux11_ia64`	`vendor`** (default), `ethernet`
IBM	64-bit	`aix51_64`	`vendor`** (default), `ethernet`

(*) Node processes on the same machine communicate by shared memory. ANSYS FLUENT lets the MPI autoselect the best interconnect available on the system. Users can specify an interconnect to override that selection. Ethernet is the fallback choice.

(**) `vendor` indicates a proprietary vendor interconnect. The specific proprietary interconnects that are supported are dictated by those that the vendor's MPI supports.

Table 34.5 Available MPIs for Linux/UNIX Platforms

MPI	Syntax (flag)	Communication Library	Notes
`hp`	`-mpi=hp`	HP MPI	General purpose for SMPs and clusters
`intel`	`-mpi=intel`	Intel MPI	General purpose for SMPs and clusters
`mpich2`	`-mpi=mpich2`	MPICH2	For both SMPs and Ethernet clusters (UNIX only)
`sgi`	`-mpi=sgi`	SGI MPI for Altix	Only for SGI Altix systems (SMP); must start ANSYS FLUENT on a system where parallel node processes are to run
`vendor`	`-mpi=vendor`	Vendor MPI	
`openmpi`	`-mpi=openmpi`	Open MPI	Open source MPI-2 implementation. For both SMPs and clusters.

Table 34.6 Supported MPIs for Linux/UNIX Architectures (Per Interconnect)

Architecture	Ethernet	Myrinet*	Infiniband	Proprietary Systems
`lnx86`	`hp` (default), `mpich2`	`hp`	`hp`	-
`lnamd64`	`hp` (default), `intel`	`hp` (default), `openmpi`	`hp` (default), `intel`, and `openmpi`	-
`lnia64`	`hp` (default), `intel`	`hp`	`hp` (default), `intel`	sgi [for `-paltix`]
`aix51_64`	`vendor` (default), `openmpi`	-	-	vendor [for `-pvendor`]

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Architecture	Ethernet	Myrinet*	Infiniband	Proprietary Systems
hpux11_ia64	vendor (default)	-	-	vendor [for -pvendor]
ultra_64	vendor (default)	-	-	vendor [for -pvendor]

(*) Both MX and GM Myrinet protocols are supported. ANSYS FLUENT will automatically detect which type is running on the system and will use that particular protocol. You only have to supply the -pmyrinet option. If the hardware supports it, the installation and usage of Myrinet MX is recommended (please consult your Myrinet vendor for applicability).

34.4.2. Setting Up Your Remote Shell and Secure Shell Clients

For cluster computing on Linux or UNIX systems, most parallel versions of ANSYS FLUENT will need the user account set up such that you can connect to all nodes on the cluster (using either the remote shell (rsh) client or the secure shell (ssh) client) without having to enter a password each time for each machine.

Provided that the appropriate server daemons (either rshd or sshd) are running, this section briefly describes how you can configure your system in order to use ANSYS FLUENT for parallel computing.

34.4.2.1. Configuring the rsh Client

The remote shell client (rsh), is widely deployed and used. It is generally easy to configure, and involves adding all the machine names, each on a single line, to the .rhosts file in your home directory.

If you refer to the machine you are currently logged on as the 'client', and if you refer to the remote machine to which you seek password-less login as the 'server', then on the server, you can add the name of your client machine to the .rhosts file. The name could be a local name or a fully qualified name with the domain suffix. Similarly, you can add other clients from which you require similar access to this server. These machines are then "trusted" and remote access is allowed without the further need for a password. This setup assumes you have the same user ID on all the machines. Otherwise, each line in the .rhosts file would need to contain the machine name as well as the user ID for the client that you want access to. Please refer to your system documentation for further usage options.

Note that for security purposes, the .rhosts file must be readable only by the user.

34.4.2.2. Configuring the ssh Client

The secure shell client (ssh), is a more secure alternative than rsh and is also used widely. Depending on the specific protocol and the version deployed, configuration involves a few steps. SSH1 and SSH2 are two current protocols. Open SSH is an open implementation of the SSH2 protocol and is backwards compatible with the SSH1 protocol. To add a client machine, with respect to user configuration, the following steps are involved:

1. Generate a public-private key pair using ssh-keygen (or using a graphical user interface client). For example:

   ```
   % ssh-keygen -t dsa
   ```

   where it creates a Digital Signature Authority (DSA) type key pair.

2. Place your public key on the remote host.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For SSH1, insert the contents of the client (`~/.ssh/identity.pub`) into the server (`~/.ssh/authorized_keys`).
- For SSH2, insert the contents of the client (`~/.ssh/id_dsa.pub`) into the server (`~/.ssh/authorized_keys2`).

The client machine is now added to the access list and you no longer need to type in a password each time. For additional information, consult your system administrator or refer to your system documentation.

34.5. Mesh Partitioning and Load Balancing

Information about mesh partitioning and load balancing is provided in the following sections:

34.5.1. Overview of Mesh Partitioning
34.5.2. Partitioning the Mesh Automatically
34.5.3. Partitioning the Mesh Manually and Balancing the Load
34.5.4. Using the Partitioning and Load Balancing Dialog Box
34.5.5. Mesh Partitioning Methods
34.5.6. Checking the Partitions
34.5.7. Load Distribution
34.5.8. Troubleshooting

34.5.1. Overview of Mesh Partitioning

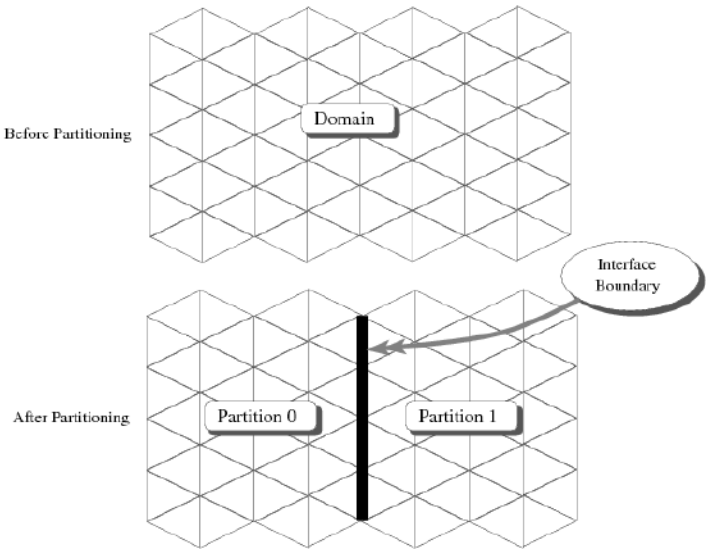
When you use the parallel solver in ANSYS FLUENT, you need to partition or subdivide the mesh into groups of cells that can be solved on separate processors (see *Figure 34.6* (p. 1604)). You can either use the automatic partitioning algorithms when reading an unpartitioned mesh into the parallel solver (recommended approach, described in *Partitioning the Mesh Automatically* (p. 1604)), or perform the partitioning yourself in the serial solver or after reading a mesh into the parallel solver (as described in *Partitioning the Mesh Manually and Balancing the Load* (p. 1605)). In either case, the available partitioning methods are those described in *Mesh Partitioning Methods* (p. 1616). You can partition the mesh before or after you set up the problem (by defining models, boundary conditions, etc.).

Note that the relative distribution of cells among compute nodes will be maintained during mesh adaption, so repartitioning after adaption is not required. For details, see *Load Distribution* (p. 1624).

If you use the serial solver to set up the problem before partitioning, the machine on which you perform this task must have enough memory to read in the mesh. If your mesh is too large to be read into the serial solver, you can read the unpartitioned mesh directly into the parallel solver (using the memory available in all the defined hosts) and have it automatically partitioned. In this case you will set up the problem after an initial partition has been made. You will then be able to manually repartition the case if necessary. See *Partitioning the Mesh Automatically* (p. 1604) and *Partitioning the Mesh Manually and Balancing the Load* (p. 1605) for additional details and limitations, and *Checking the Partitions* (p. 1622) for details about checking the partitions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.6 Partitioning the Mesh



34.5.2. Partitioning the Mesh Automatically

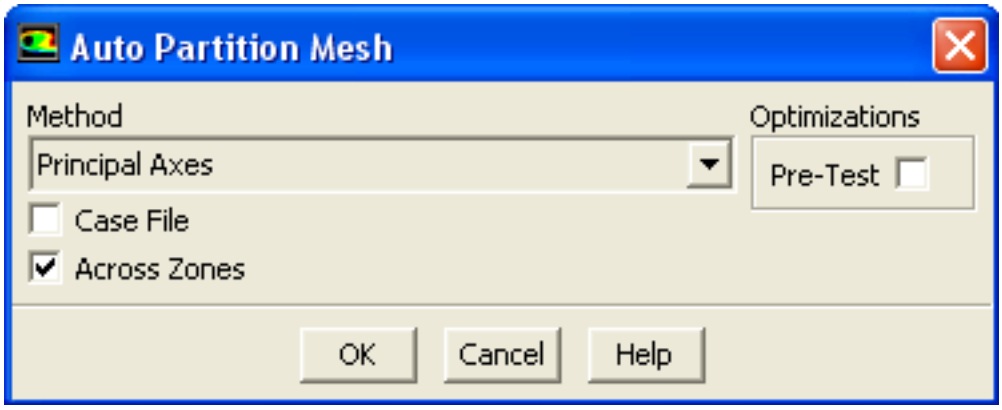
For automatic mesh partitioning, you can select the partition method and other options for creating the mesh partitions before reading a case file into the parallel version of the solver. For some of the methods, you can perform pretesting to ensure that the best possible partition is performed. See *Mesh Partitioning Methods* (p. 1616) for information about the partitioning methods available in ANSYS FLUENT.

The procedure for partitioning automatically in the parallel solver is as follows:

1. (optional) Set the partitioning parameters in the **Auto Partition Mesh** dialog box (*Figure 34.7* (p. 1604)).

   Parallel → **Auto Partition...**

   Figure 34.7 The Auto Partition Mesh Dialog Box



   If you are reading in a mesh file or a case file for which no partition information is available, and you keep the **Case File** option turned on, ANSYS FLUENT will partition the mesh using the method displayed in the **Method** drop-down list.

   If you want to specify the partitioning method and associated options yourself, the procedure is as follows:

   a. Turn off the **Case File** option. The other options in the dialog box will become available.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the partition method in the **Method** drop-down list. The choices are the techniques described in *Partition Methods* (p. 1616).

c. You can choose to independently apply partitioning to each cell zone, or you can allow partitions to cross zone boundaries using the **Across Zones** check button. It is recommended that you *not* partition cells zones independently (by turning off the **Across Zones** check button) unless cells in different zones will require significantly different amounts of computation during the solution phase (e.g., if the domain contains both solid and fluid zones).

d. If you have chosen the **Principal Axes** or **Cartesian Axes** method, you can improve the partitioning by enabling the automatic testing of the different bisection directions before the actual partitioning occurs. To use pretesting, turn on the **Pre-Test** option. Pretesting is described in *Pretesting* (p. 1621).

e. Click **OK**.

If you have a case file where you have already partitioned the mesh, and the number of partitions divides evenly into the number of compute nodes, you can keep the default selection of **Case File** in the **Auto Partition Mesh** dialog box. This instructs ANSYS FLUENT to use the partitions in the case file.

2. Read the case file.

File → **Read** → **Case...**

34.5.2.1. Reporting During Auto Partitioning

As the mesh is automatically partitioned, some information about the partitioning process will be displayed in the console. If you want additional information, you can display a report from the **Partitioning and Load Balancing** dialog box after the partitioning is completed.

Parallel → **Partitioning and Load Balancing...**

When you click the **Print Active Partitions** or **Print Stored Partitions** button in the **Partitioning and Load Balancing** dialog box, ANSYS FLUENT will display the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active or stored partition in the console. In addition, it will display the minimum and maximum cell, face, interface, and face-ratio variations. For details, see *Interpreting Partition Statistics* (p. 1622). You can examine the partitions graphically by following the directions in *Checking the Partitions* (p. 1622).

34.5.3. Partitioning the Mesh Manually and Balancing the Load

Automatic partitioning in the parallel solver (described in *Partitioning the Mesh Automatically* (p. 1604)) is the recommended approach to mesh partitioning, but it is also possible to partition the mesh manually in either the serial solver or the parallel solver. After automatic or manual partitioning, you will be able to inspect the partitions created (for details, see *Checking the Partitions* (p. 1622)) and optionally repartition the mesh, if necessary. Again, you can do so within the serial or the parallel solver, using the **Partitioning and Load Balancing** dialog box. A partitioned mesh may also be used in the serial solver without any loss in performance.

34.5.3.1. Guidelines for Partitioning the Mesh

The following steps are recommended for partitioning a mesh manually:

1. Partition the mesh using the default bisection method (**Principal Axes**) and optimization (**Smooth**).
2. Examine the partition statistics, which are described in *Interpreting Partition Statistics* (p. 1622). Your aim is to achieve small values of `Interface ratio variation` and `Global interface ratio`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

while maintaining a balanced load (`Cell variation`). If the statistics are not acceptable, try one of the other partition methods.

3. Once you determine the best partition method for your problem, you can turn on **Pre-Test** (for details, see *Pretesting* (p. 1621)) to improve it further, if desired.
4. You can also improve the partitioning using the **Merge** optimization, if desired.

Instructions for manual partitioning are provided below.

34.5.4. Using the Partitioning and Load Balancing Dialog Box

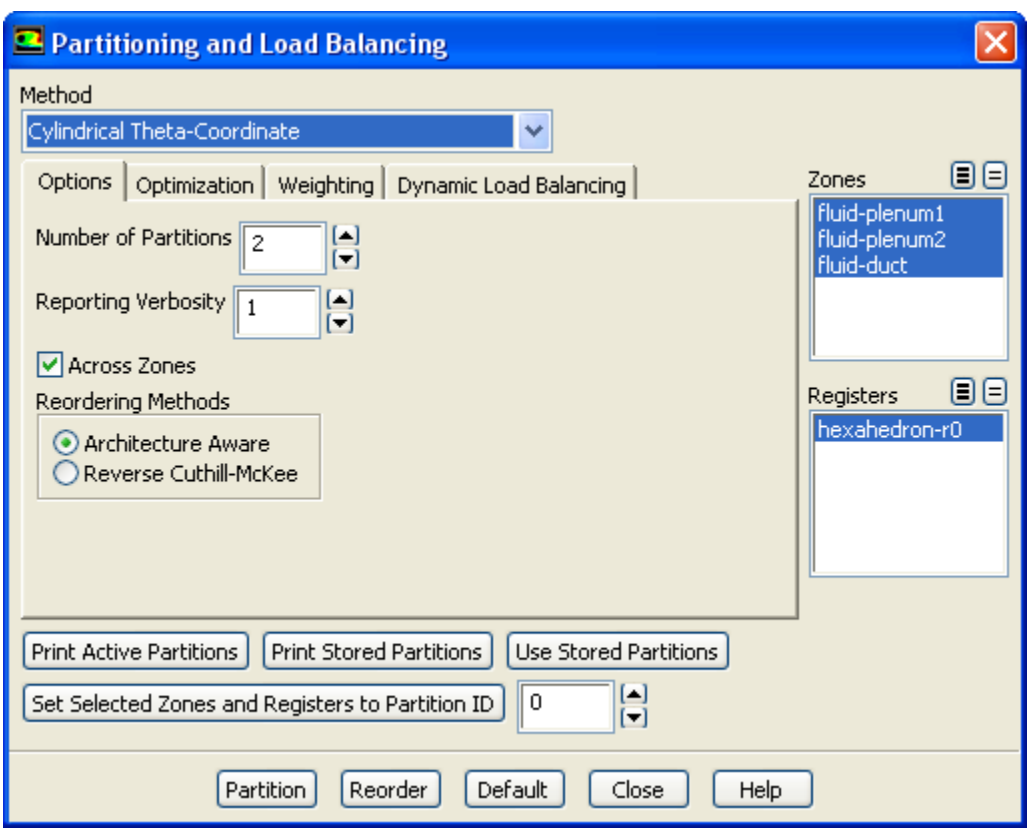
34.5.4.1. Partitioning

In order to partition the mesh, you need to select the partition method for creating the mesh partitions, set the number of partitions, select the zones and/or registers, and choose the optimizations to be used. For some methods, you can also perform pretesting to ensure that the best possible partition is performed. Once you have set all the parameters in the **Partitioning and Load Balancing** dialog box to your satisfaction, click the **Partition** button to subdivide the mesh into the selected number of partitions using the prescribed method and optimization(s). For recommended partitioning strategies see *Guidelines for Partitioning the Mesh* (p. 1605).

You can set the relevant inputs in the **Partitioning and Load Balancing** dialog box (*Figure 34.8* (p. 1606) in the parallel solver, or *Figure 34.9* (p. 1607) in the serial solver) in the following manner:

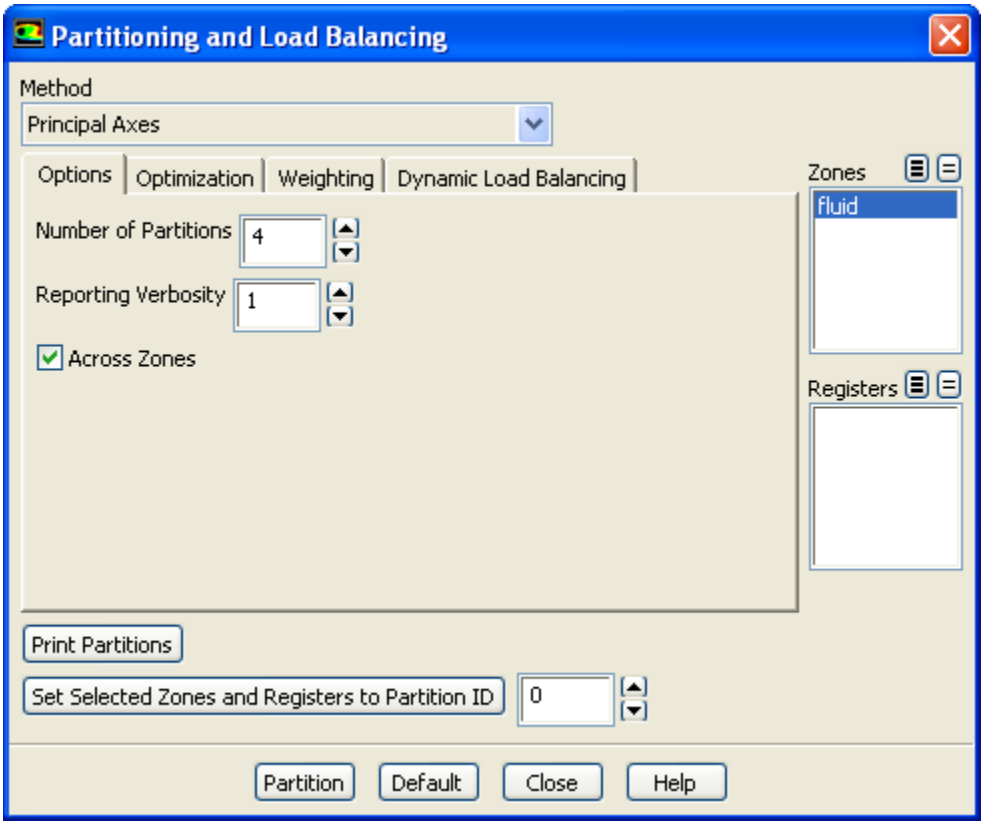
Parallel → Partitioning and Load Balancing...

Figure 34.8 The Partitioning and Load Balancing Dialog Box in the Parallel Solver



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.9 The Partitioning and Load Balancing Dialog Box in the Serial Solver



1. Select the **Method** from the drop-down list. The choices are described in *Partition Methods* (p. 1616).
2. In the **Options** tab
   a. Set the desired number of mesh partitions in the **Number of Partitions** field. You can use the counter arrows to increase or decrease the value, instead of typing in the box. The number of mesh partitions must be an integer number which is divisible by the number of processors available for parallel computing.
   b. Set the **Reporting Verbosity**. This allows you to control what is displayed in the console. For details, see *Reporting During Partitioning* (p. 1613).
   c. You can choose to independently apply partitioning to each cell zone, or you can allow partitions to cross zone boundaries using the **Across Zones** check button. It is recommended that you *not* partition cell zones independently (by turning off the **Across Zones** check button) unless cells in different zones will require significantly different amounts of computation during the solution phase (e.g., if the domain contains both solid and fluid zones).
   d. Select the **Reordering Method** for partitions to optimize parallel performance:
      - **Architecture Aware**: This is the default option and it accounts for the system architecture and network topology in remapping the partitions to the processors.
      - **Reverse Cuthill-McKee**: This option minimizes the bandwidth of the compute-node connectivity matrix (the maximum distance between two connected processes) without incorporating the system architecture.

      The reordering methods are parallel performance tuning options. After the case is initially partitioned for parallel processing, the partition reordering step will remap the partitions in a more optimal way to improve parallel performance.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Some restrictions apply for the Architecture-aware reordering method:

- The method is not applicable when only a single machine is used for the simulation.
- The machines in the cluster should be uniform in the sense that they must all be of the same architecture (dual/quad, core, etc.), and all machines must be uniformly loaded, i.e., carry the same number of ANSYS FLUENT node processes.

After initially loading the case into a parallel session, you can click the **Reorder** button to reorder the partitions. The necessary algorithms are executed and ANSYS FLUENT will report if it can find a more optimal mapping for the partitions, as well as the percentage of improvement in inter-machine communications. If the reported improvement is significant (say, more than 5%), then you can click the **Use Stored Partitions** button to use the new partition mapping. This will generally entail large data transfers amongst all the processes, and another reliable method to activate the new partitions would be to write out a case file and load it back in to a new parallel session. The process is similar to re-partitioning with a new partitioning method, for example. Note that sometimes, depending on the cluster configuration and initial case partitioning, and if the partitions have already been reordered, no improvement is possible, and this will be reported in the console, after clicking the **Reorder** button. You can simply continue in this case, and there will be no effect on the simulation. Also, note that partition reordering is specific to the current parallel configuration and should be repeated if the number of machines used changes during subsequent computations.

3. In the **Optimization** tab

   a. You can activate and control the desired optimization methods (described in *Optimizations* (p. 1620)). You can activate the **Merge** and **Smooth** schemes by enabling the check button next to each one. For each scheme, you can also set the number of **Iterations**. Each optimization scheme will be applied until appropriate criteria are met, or the maximum number of iterations has been executed. If the **Iterations** counter is set to 0, the optimization scheme will be applied until completion, with no limit on maximum number of iterations.

   b. Choosing the **Principal Axes** or **Cartesian Axes** method, you can improve the partitioning by enabling the automatic testing of the different bisection directions before the actual partitioning occurs. To use pretesting, enable the **Pre-Test** option. Pretesting is described in *Pretesting* (p. 1621).

4. In the **Zones** and/or **Registers** lists, select the zone(s) and/or register(s) for which you want to partition. For most cases, you will select all **Zones** (the default) to partition the entire domain. See below for details.

5. You can assign selected **Zones** and/or **Registers** to a specific partition ID by entering a value for the **Set Selected Zones and Registers to Partition ID**. For example, if the **Number** of partitions for your mesh is 2, then you can only use IDs of `0` or `1`. If you have three partitions, then you can enter IDs of `0`, `1`, or `2`. This can be useful in situations where the gradient at a region is known to be high. In such cases, you can mark the region or zone and set the marked cells to one of the partition IDs, thus preventing the partition from going through that region. This in turn will facilitate convergence. This is also useful in cases where mesh manipulation tools are not available in parallel. In this case, you can assign the related cells to a particular ID so that the mesh manipulation tools are now functional.

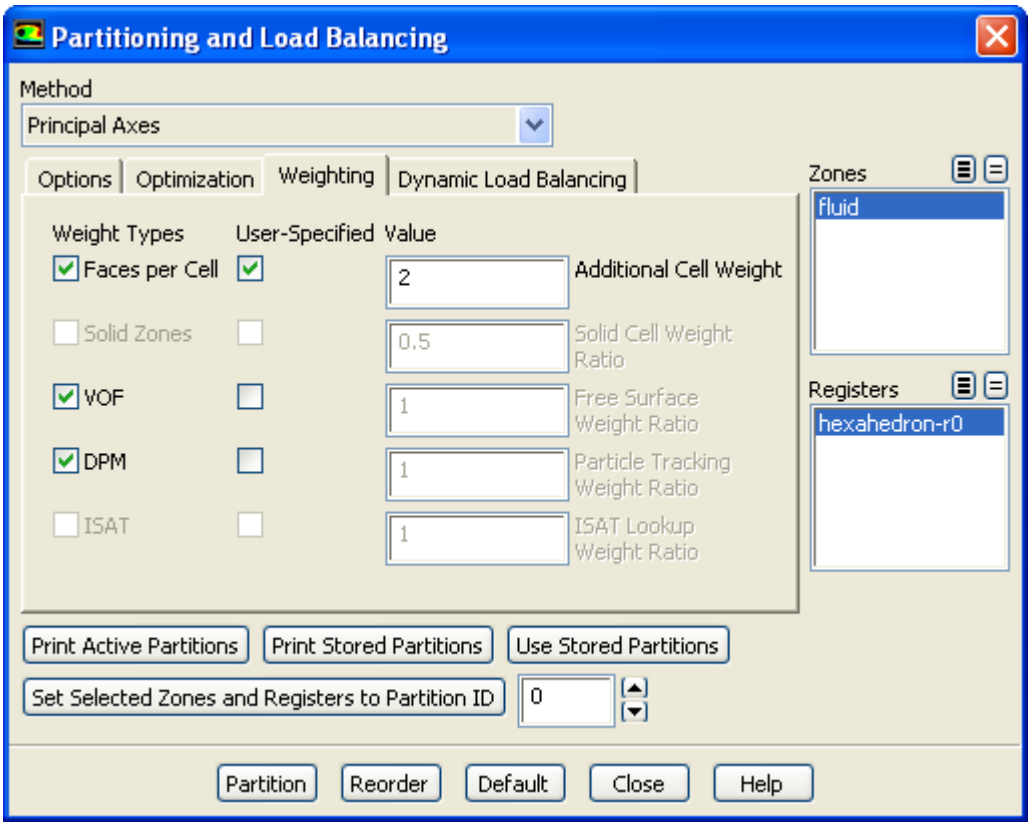
   If you are running the parallel solver, and you have marked your region and assigned an ID to the selected **Zones** and/or **Registers**, click the **Use Stored Partitions** button to make the new partitions valid.

   Refer to the example described later in this section for a demonstration of how selected registers are assigned to a partition (*Example of Setting Selected Registers to Specified Partition IDs* (p. 1611)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the **Weighting** tab (*Figure 34.10* (p. 1609)), you can set the appropriate weights, prior to partitioning the mesh, to improve load balancing and overall performance. You can control weights for cells, solid zones, VOF, and DPM. You can either rely on ANSYS FLUENT timers to set the weight scaling, or you can specify the value by enabling **User-Specified**.

Figure 34.10 The Weighting Tab in the Partitioning and Load Balancing Dialog Box for the Parallel Solver



a. Enable **Faces per Cell** so that the partitioning assigns a weight to each cell based on its number of faces. This type of weighting is advantageous when the case has mixed or polyhedral cell zones. If you enable the **User-Specified** check box, the weight assigned to each cell will be the number of faces plus the **Additional Cell Weight** you enter in the number-entry box under **Value**. By default the **Faces per Cell** weighting is enabled with the **Additional Cell Weight** set to **2**.

b. Enable **Solid Zones** weighting to allow the partitioning to take solid zones into consideration. By default, ANSYS FLUENT will scale the weight based on the computational time of the solid and fluid zones. If an iteration has not been run, then you may need to specify a value after enabling the user-specified check box. The value you enter is relative to the fluid weight. Typically, it should be less than `1` (e.g., `0.0001`) so that the calculation will be quicker for the solid zone compared to the fluid zone. Entering a value greater than `1` for the **Solid Zones** means that the calculation will take longer and will be more computationally expensive for the solid zone compared to the fluid zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Setting the **Solid Zones** either too high or too low can cause a load imbalance, depending on which equations are being solved. If you are not sure what would be an appropriate value, have ANSYS FLUENT automatically set it for you by making sure that **User-Specified** is disabled.

c. Enable **VOF** weighting to allow the partitioning to consider the imbalance caused by the free surface reconstruction with the geo-reconstruct scheme. Therefore, it is only available when using the VOF model with geometric reconstruction. You may use the user-specified value before timers are collected, or if you want to specify a value other than timing statistics. The specified value is the VOF proportion of the total computational effort.

d. Enable **DPM** weighting to set the weight of DPM particles relative to the continuous phase. DPM weights are valid when you have particle tracking in your simulation, where the user-specified value is the DPM proportion of the total computational effort relative to the continuous phase. Note that this is only available when you have injections defined. For details, see *Modeling Discrete Phase* (p. 1007).

   The DPM weight takes into account the distribution of the tracking effort over the partitions and it is available after at least one calculation step with particle tracking. Displaying Particle Tracks does not change the weights. The computational effort is determined by the number of DPM steps performed in each cell. This weight becomes more important when the time for the particle tracking of particles exceeds the time for solving the flow. Enabling this option in the **Weighting** tab activates the counting of the particle steps in the cells. These values are available for contour and vector plots when using the **Discrete Phase Model** and **DPM Steps per Cell** variable. Additionally, it is preferable to partition along the main path of the particles, which avoids crossing of partitions and communication overhead. The particular partitioning algorithm avoids the partition interfaces on the faces with high numbers of particle paths crossing. After repartitioning, the DPM weights are reset before the next particle tracking.

e. Enable **ISAT** weighting to balance the load during the ISAT table lookup for the stiff-chemistry Laminar, EDC or PDF Transport models. The ISAT algorithm builds an unstructured table in N species dimensions for storage and retrieval of the chemistry mappings. Since chemistry is usually computationally expensive, this storage/retrieval can be very time-consuming (for information about ISAT, refer to In-Situ Adaptive Tabulation (ISAT) in the Theory Guide). Each parallel node builds its own table, and there is no message passing to tables on other nodes. As some nodes may have more chemical reactions than others (for example one parallel node may contain just air at a constant temperature, in which case the ISAT table will contain only one entry and calculation will be rapid), there may be a load imbalance. The dynamic load balancing algorithm will migrate cells from high computational load nodes to low computational load nodes.

   If you decide to specify a value, this user-specified value is the ISAT proportion of the total computational effort.

7. When using the dynamic mesh model in your parallel simulations, the **Partition** dialog box includes an **Auto Repartition** option and a **Repartition Interval** setting. These parallel partitioning options are provided because ANSYS FLUENT migrates cells when local remeshing and smoothing are performed. Therefore, the partition interface becomes very wrinkled and the load balance may deteriorate. By default, the **Auto Repartition** option is selected, where a percentage of interface faces and loads are automatically traced. When this option is selected, ANSYS FLUENT automatically determines the most appropriate repartition interval based on various simulation parameters. Sometimes, using the **Auto Repartition** option provides insufficient results, therefore, the **Repartition Interval** setting can be

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

used. The **Repartition Interval** setting lets you to specify the interval (in time steps or iterations respectively) when a repartition is enforced. When repartitioning is not desired, you can set the **Repartition Interval** to zero.

Important

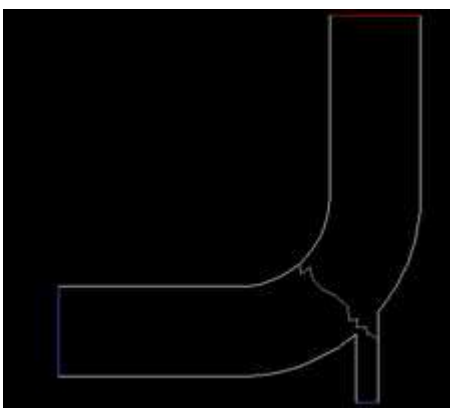
Note that when dynamic meshes and local remeshing is utilized, updated meshes may be slightly different in parallel ANSYS FLUENT (when compared to serial ANSYS FLUENT or when compared to a parallel solution created with a different number of compute nodes), resulting in very small differences in the solutions.

8. Click the **Partition** button to partition the mesh.
9. If you decide that the new partitions are better than the previous ones (if the mesh was already partitioned), click the **Use Stored Partitions** button to make the newly stored cell partitions the active cell partitions. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

34.5.4.1.1. Example of Setting Selected Registers to Specified Partition IDs

1. Start ANSYS FLUENT in parallel. The case in this example was partitioned across two nodes.
2. Read in your case.
3. Display the mesh with the **Partitions** option enabled in the **Mesh Display** dialog box (*Figure 34.11* (p. 1611)).

Figure 34.11 The Partitioned Mesh



4. Adapt your region and mark your cells. (For details, see *Performing Region Adaption* (p. 1336)). This creates a register.
5. Open the **Partitioning and Load Balancing** dialog box.
6. Set the **Set Selected Zones and Registers to Partition ID** to `0` and click the corresponding button. This displays the following output in the ANSYS FLUENT console:

```
>> 2 Active Partitions:
   ----------------------------------------------------------------------
   Collective Partition Statistics:       Minimum       Maximum    Total
   ----------------------------------------------------------------------
   Cell count                                 459           459        918
   Mean cell count deviation                0.0%          0.0%
   Partition boundary cell count           11            11          22
   Partition boundary cell count ratio    2.4%          2.4%       2.4%
   Face count                                 764          1714        2461
```

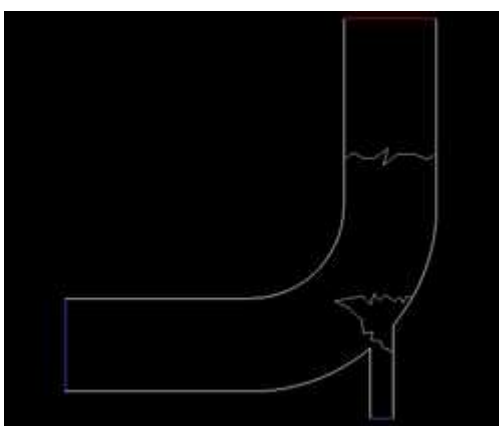
```
  Mean face count deviation                -38.3%        38.3%
  Partition boundary face count           13            13          17
  Partition boundary face count ratio    0.8%          1.7%       0.7%
  Partition neighbor count                 1             1
  ---------------------------------------------------------------------
  Partition Method                          Principal Axes
  Stored Partition Count                   2
Done.
```

7. Click the **Use Stored Partitions** button to make the new partitions valid. This migrates the partitions to the compute-nodes. The following output is then displayed in the ANSYS FLUENT console:

```
Migrating partitions to compute-nodes.
>> 2 Active Partitions:
      P    Cells   I-Cells   Cell Ratio    Faces   I-Faces   Face Ratio   Neighbors
      0      672       24         0.036     2085        29       0.014            1
      1      246       24         0.098      425        29       0.068            1
  ---------------------------------------------------------------------
  Collective Partition Statistics:      Minimum        Maximum       Total
  ---------------------------------------------------------------------
  Cell count                                 246             672           918
  Mean cell count deviation               -46.4%          46.4%
  Partition boundary cell count          24               24             48
  Partition boundary cell count ratio   3.6%            9.8%           5.2%
  Face count                                 425            2085          2461
  Mean face count deviation               -66.1%          66.1%
  Partition boundary face count          29               29             49
  Partition boundary face count ratio   1.4%            6.8%           2.0%
  Partition neighbor count                1                1
  ---------------------------------------------------------------------
  Partition Method                          Principal Axes
  Stored Partition Count                  2
Done.
```

8. Display the mesh (*Figure 34.12* (p. 1612)).

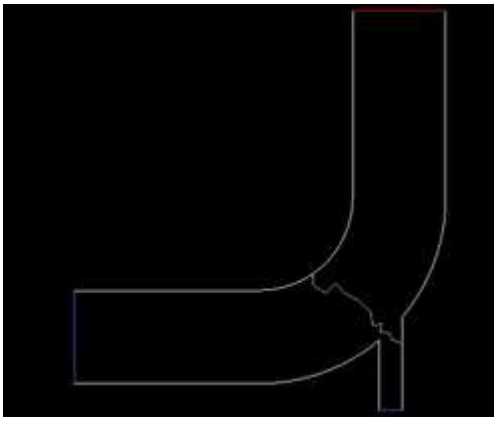
Figure 34.12 The Partitioned ID Set to Zero



9. This time, set the **Set Selected Zones and Registers to Partition ID** to 1 and click the corresponding button. This displays a report in the ANSYS FLUENT console.
10. Click the **Use Stored Partitions** button to make the new partitions valid and to migrate the partitions to the compute-nodes.
11. Display the mesh (*Figure 34.13* (p. 1613)). Notice now that the partition appears in a different location as specified by your partition ID.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.13 The Partitioned ID Set to 1



Important

Although this example demonstrates setting selected registers to specific partition IDs in parallel, it can be similarly applied in serial.

34.5.4.1.2. Partitioning Within Zones or Registers

The ability to restrict partitioning to cell zones or registers gives you the flexibility to apply different partitioning strategies to subregions of a domain. For example, if your geometry consists of a cylindrical plenum connected to a rectangular duct, you may want to partition the plenum using the **Cylindrical Axes** method, and the duct using the **Cartesian Axes** method.

If the plenum and the duct are contained in two different cell zones, you can select one at a time and perform the desired partitioning, as described in *Using the Partitioning and Load Balancing Dialog Box* (p. 1606). If they are not in two different cell zones, you can create a cell register (basically a list of cells) for each region using the functions that are used to mark cells for adaption. These functions allow you to mark cells based on physical location, cell volume, gradient or isovalue of a particular variable, and other parameters. See *Adapting the Mesh* (p. 1325) for information about marking cells for adaption. *Manipulating Adaption Registers* (p. 1343) provides information about manipulating different registers to create new ones. Once you have created a register, you can partition within it as described in *Example of Setting Selected Registers to Specified Partition IDs* (p. 1611).

Important

Note that partitioning within zones or registers is not available when **Metis** is selected as the partition **Method**.

For dynamic mesh applications, ANSYS FLUENT stores the partition method used to partition the respective zone. Therefore, if repartitioning is done, ANSYS FLUENT uses the same method that was used to partition the mesh.

34.5.4.1.3. Reporting During Partitioning

As the mesh is partitioned, information about the partitioning process will be displayed in the console. By default, the number of partitions created, the number of bisections performed, the time required for the partitioning, and the minimum and maximum cell, face, interface, and face-ratio variations will be displayed. (For details, see *Interpreting Partition Statistics* (p. 1622).) If you increase the **Reporting Verbosity** to 2 from

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the default value of 1, the partition method used, the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each partition will also be displayed in the console. If you decrease the **Reporting Verbosity** to 0, only the number of partitions created and the time required for the partitioning will be reported.

You can request a portion of this report to be displayed again after the partitioning is completed. When you click the **Print Active Partitions** or **Print Stored Partitions** button in the parallel solver, ANSYS FLUENT will display the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active or stored partition in the console. In addition, it will display the minimum and maximum cell, face, interface, and face-ratio variations. In the serial solver, you will obtain the same information about the stored partition when you click **Print Partitions**. For details, see *Interpreting Partition Statistics* (p. 1622).

Important

Recall that to make the stored cell partitions the active cell partitions you must click the **Use Stored Partitions** button. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

34.5.4.1.4. Resetting the Partition Parameters

If you change your mind about your partition parameter settings, you can easily return to the default settings assigned by ANSYS FLUENT by clicking on the **Default** button. When you click the **Default** button, it will become the **Reset** button. The **Reset** button allows you to return to the most recently saved settings (i.e., the values that were set before you clicked on **Default**). After execution, the **Reset** button will become the **Default** button again.

34.5.4.2. Load Balancing

A dynamic load balancing capability is available in ANSYS FLUENT. The principal reason for using parallel processing is to reduce the turnaround time of your simulation, which may be achieved by the following means:

- Faster machines, e.g., faster CPU, memory, cache, and, transportation speed between the CPU and memory
- Faster interconnects, e.g., smaller latency and larger bandwidth
- Better Load balancing, e.g., load is evenly distributed and CPUs are not idled during calculation

The first two are evolving with technology advancement, which is beyond the scope of this document. The third item is regarding optimization of available computation power. Here we are mainly talking about load balancing on dedicated homogeneous resources, which is often the case nowadays. If you are not using a dedicated homogeneous resource, you may need to account for differences in CPU speeds during partitioning by specifying a load distribution. (For details, see *Load Distribution* (p. 1624)).

On a dedicated homogeneous system, the key for load balancing is how to evaluate the computational requirement of each cell. By default, ANSYS FLUENT assumes that each cell requires the same computational work, but this is often not the case. For example

- A hexahedral cell demands more CPU and memory than a tetrahedral cell.
- A cell with particle tracking will use more time than a cell without particle tracking.
- ISAT species model cells may have magnitude differences in time usage.

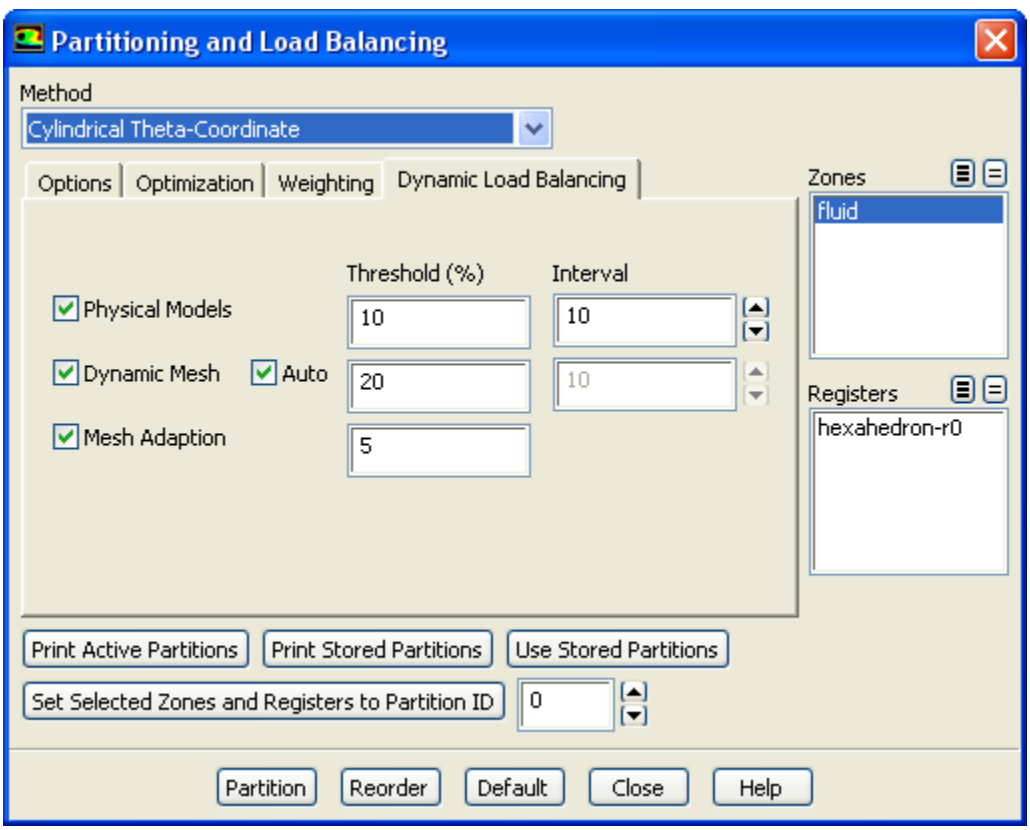
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To balance these differences, ideally, the time used in each cell can be recorded and load balancing can be achieved based on these time statistics. But in reality, this is hardly possible, because the statistics itself may require more time than the real computation effort. In addition, accuracy of timers is questionable in highly granulized accounting. Instead, we identify features causing computation unevenness and try to record time usage for these features in a collective manner. For a more detailed description of this, please refer to *Partitioning* (p. 1606) in the discussion of the **Weighting** tab. In addition, the imbalance may be in a dynamic manner, for example

- The mesh may be changed by adaption or mesh movement.
- In unsteady cases, particle tracking may move from one region to another region.

The current dynamic load balancing has been implemented for better scalability of cases with imbalanced physical or geometrical models, thus reducing the simulation time. The implementation is based on considering weights from these models scaled by CPU time usage. Load balancing for DPM, VOF, cell type (number of faces per cell), and solid zones can be performed. In addition, cell weight based load balancing and machine load distribution can also be specified. (For details, see *Load Distribution* (p. 1624)). ANSYS FLUENT takes the weights from physical models and considers them for partitioning. The weights are assembled based on the time used by each physical model. For dynamic load balancing, the load is checked and balanced based on your specified imbalance threshold. To apply dynamic load balancing on the various models, click the **Dynamic Load Balancing** tab and select the required balancing as follows:

Figure 34.14 The Dynamic Load Balancing Tab



1. Enable **Physical Models** load balancing during iterations so that the load will be evaluated for time usage and weight distribution, based on the **Interval** that you provide. If the imbalance exceeds the specified **Threshold**, then repartitioning will be performed by considering the selected weights. **Physical Models** load balancing will only be available when you have the specific physical models

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enabled in the case. You will be prompted to enable the weights for those models. When weights for the physical models are all disabled, you will be prompted to disable **Physical Models** load balancing.

2. Enable **Dynamic Mesh** if there is any dynamic mesh movement. Load balancing, based on the number of cells, will be checked and balanced if the imbalance threshold is exceeded. These parallel partitioning options are provided because ANSYS FLUENT migrates cells when local remeshing and smoothing are performed. Therefore, the partition interface becomes very wrinkled and the load balance may deteriorate. By default, the **Auto** option is selected, where a percentage of interface faces and loads are automatically traced. When this option is selected, ANSYS FLUENT automatically determines the most appropriate repartition interval based on various simulation parameters. Sometimes, using the **Auto** option provides insufficient results; therefore, the **Interval** setting can be used. The **Interval** setting lets you specify the interval (in time steps or iterations, respectively) when load balancing is enforced. When load balancing is not desired, you may disable **Dynamic Mesh** load balancing. Dynamic Mesh load balancing is only available when you have dynamic models enabled in your case.

   Important

   Note that when dynamic meshes and local remeshing are utilized, updated meshes may be slightly different in parallel ANSYS FLUENT (when compared to serial ANSYS FLUENT or when compared to a parallel solution created with a different number of compute nodes), resulting in very small differences in the solutions.

3. Enable **Mesh Adaption**. Any time mesh adaption occurs, load balancing, based on the number of cells, will be checked and balanced if the imbalance threshold is exceeded. If problems arise in your computations due to adaption, you can disable the load balancing for **Mesh Adaption**.

34.5.5. Mesh Partitioning Methods

Partitioning the mesh for parallel processing has three major goals:

- Create partitions with equal numbers of cells.
- Minimize the number of partition interfaces — i.e., decrease partition boundary surface area.
- Minimize the number of partition neighbors.

Balancing the partitions (equalizing the number of cells) ensures that each processor has an equal load and that the partitions will be ready to communicate at about the same time. Since communication between partitions can be a relatively time-consuming process, minimizing the number of interfaces can reduce the time associated with this data interchange. Minimizing the number of partition neighbors reduces the chances for network and routing contentions. In addition, minimizing partition neighbors is important on machines where the cost of initiating message passing is expensive compared to the cost of sending longer messages. This is especially true for workstations connected in a network.

The partitioning schemes in ANSYS FLUENT use bisection or METIS algorithms to create the partitions, but unlike other schemes which require the number of partitions to be a factor of two, these schemes have no limitations on the number of partitions. For each available processor, you will create the same number of partitions (i.e., the total number of partitions will be an integral multiple of the number of processors).

34.5.5.1. Partition Methods

The mesh is partitioned using a bisection or METIS algorithm. The selected algorithm is applied to the parent domain, and then recursively applied to the child subdomains. For example, to divide the mesh into four partitions with a bisection method, the solver will bisect the entire (parent) domain into two child domains,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and then repeat the bisection for each of the child domains, yielding four partitions in total. To divide the mesh into three partitions with a bisection method, the solver will "bisect" the parent domain to create two partitions—one approximately twice as large as the other—and then bisect the larger child domain again to create three partitions in total.

The mesh can be partitioned using one of the algorithms listed below. The most efficient choice is problem-dependent, so you can try different methods until you find the one that is best for your problem. See *Guidelines for Partitioning the Mesh* (p. 1605) for recommended partitioning strategies.

Cartesian Axes
bisects the domain based on the Cartesian coordinates of the cells (see *Figure 34.15* (p. 1618)). It bisects the parent domain and all subsequent child subdomains perpendicular to the coordinate direction with the longest extent of the active domain. It is often referred to as coordinate bisection.

Cartesian Strip
uses coordinate bisection but restricts all bisections to the Cartesian direction of longest extent of the parent domain (see *Figure 34.16* (p. 1619)). You can often minimize the number of partition neighbors using this approach.

Cartesian X-, Y-, Z-Coordinate
bisects the domain based on the selected Cartesian coordinate. It bisects the parent domain and all subsequent child subdomains perpendicular to the specified coordinate direction. (See *Figure 34.16* (p. 1619).)

Cartesian R Axes
bisects the domain based on the shortest radial distance from the cell centers to that Cartesian axis (x, y, or z) which produces the smallest interface size. This method is available only in 3D.

Cartesian RX-, RY-, RZ-Coordinate
bisects the domain based on the shortest radial distance from the cell centers to the selected Cartesian axis (x, y, or z). These methods are available only in 3D.

Cylindrical Axes
bisects the domain based on the cylindrical coordinates of the cells. This method is available only in 3D.

Cylindrical R-, Theta-, Z-Coordinate
bisects the domain based on the selected cylindrical coordinate. These methods are available only in 3D.

Metis
uses the METIS software package for partitioning irregular graphs, developed by Karypis and Kumar at the University of Minnesota and the Army HPC Research Center. It uses a multilevel approach in which the vertices and edges on the fine graph are coalesced to form a coarse graph. The coarse graph is partitioned, and then uncoarsened back to the original graph. During coarsening and uncoarsening, algorithms are applied to permit high-quality partitions. Detailed information about METIS can be found in its manual [*39*] (p. 2211).

Important

- Note that when using the socket version (`-pnet`), the METIS partitioner is not available. In this case, METIS partitioning can be obtained using the partition filter, as described below.
- If you create non-conformal interfaces, and generate virtual polygonal faces, your METIS partition can cross non-conformal interfaces by using the connectivity of the virtual polygonal faces. This improves load balancing for the parallel solver and minimizes communication by decreasing the number of partition interface cells.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Polar Axes

bisects the domain based on the polar coordinates of the cells (see *Figure 34.19* (p. 1620)). This method is available only in 2D.

Polar R-Coordinate, Polar Theta-Coordinate

bisects the domain based on the selected polar coordinate (see *Figure 34.19* (p. 1620)). These methods are available only in 2D.

Principal Axes

bisects the domain based on a coordinate frame aligned with the principal axes of the domain (see *Figure 34.17* (p. 1619)). This reduces to Cartesian bisection when the principal axes are aligned with the Cartesian axes. The algorithm is also referred to as moment, inertial, or moment-of-inertia partitioning.

This is the default bisection method in ANSYS FLUENT.

Principal Strip

uses moment bisection but restricts all bisections to the principal axis of longest extent of the parent domain (see *Figure 34.18* (p. 1620)). You can often minimize the number of partition neighbors using this approach.

Principal X-, Y-, Z-Coordinate

bisects the domain based on the selected principal coordinate (see *Figure 34.18* (p. 1620)).

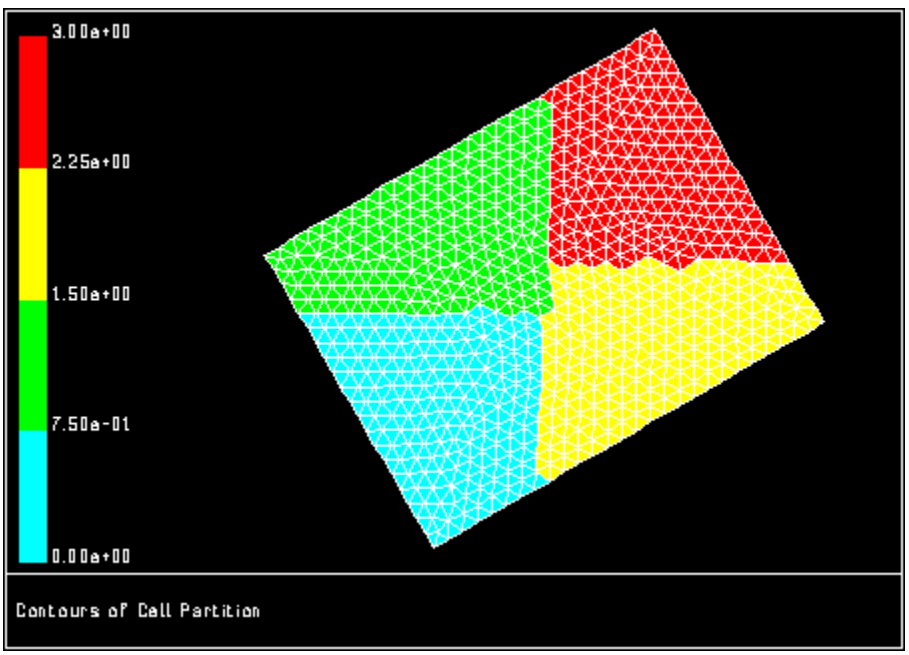
Spherical Axes

bisects the domain based on the spherical coordinates of the cells. This method is available only in 3D.

Spherical Rho-, Theta-, Phi-Coordinate

bisects the domain based on the selected spherical coordinate. These methods are available only in 3D.

Figure 34.15 Partitions Created with the Cartesian Axes Method



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.16 Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method

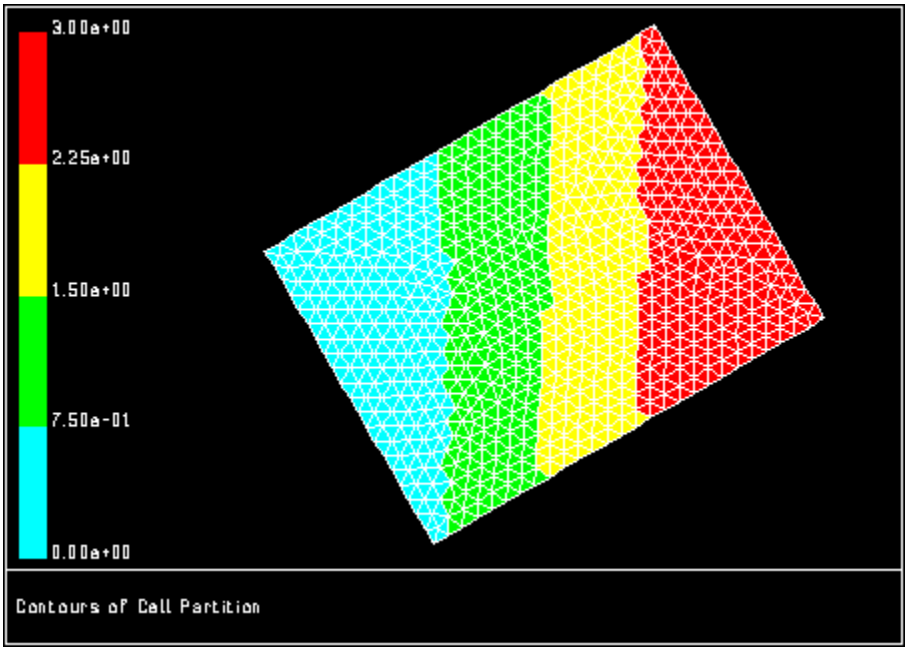
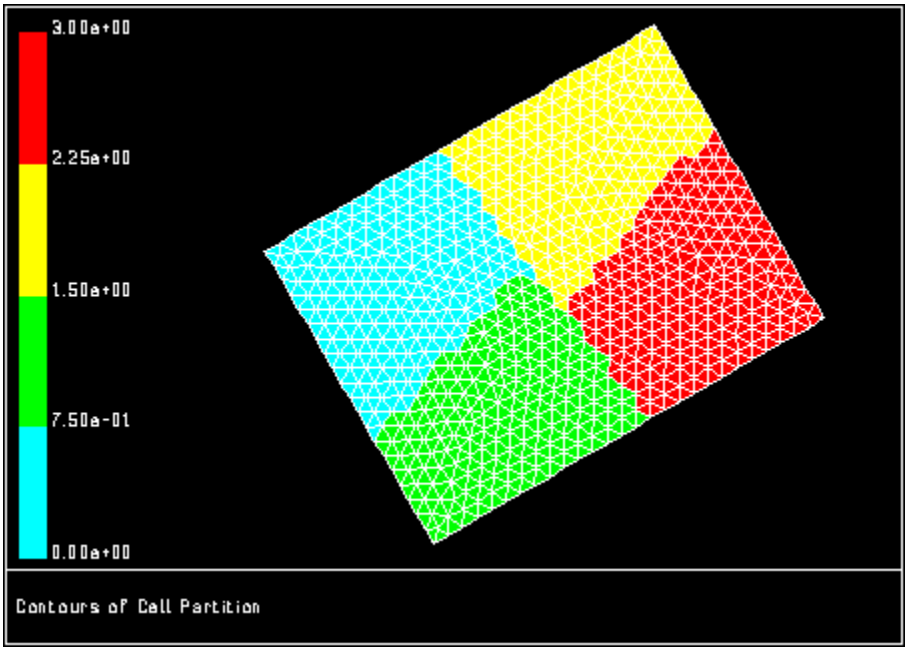


Figure 34.17 Partitions Created with the Principal Axes Method



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.18 Partitions Created with the Principal Strip or Principal X-Coordinate Method

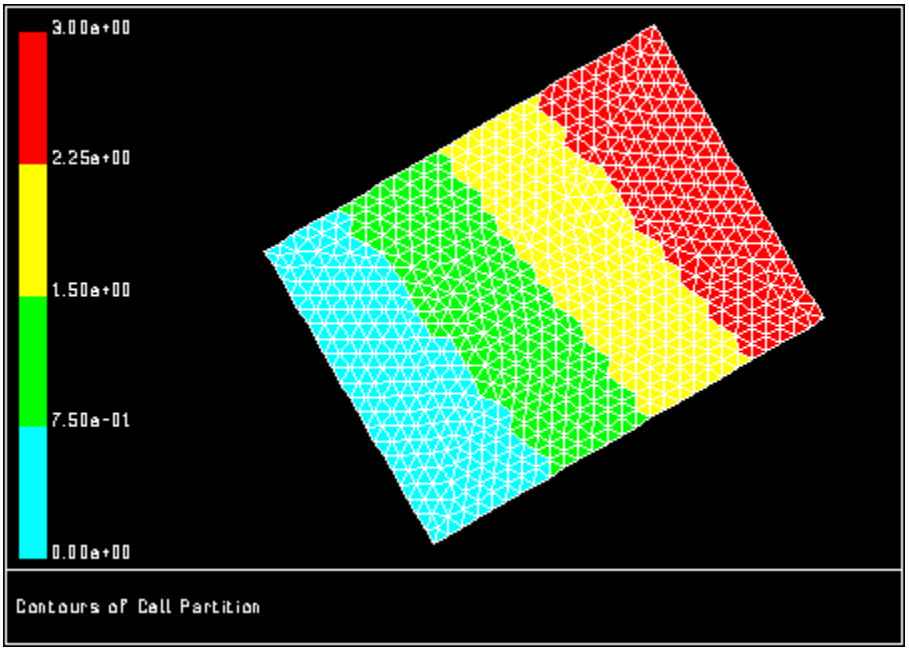
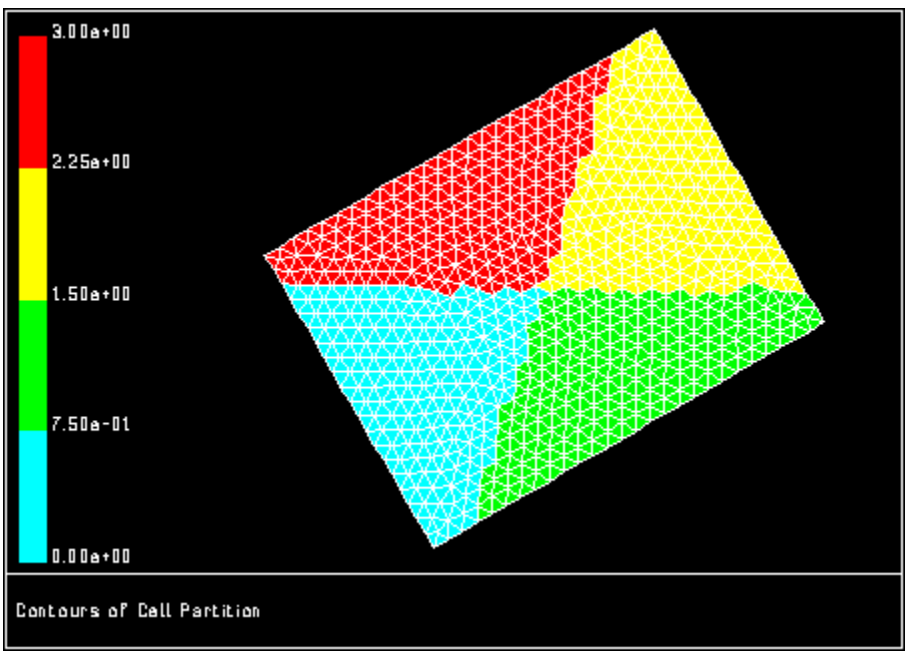


Figure 34.19 Partitions Created with the Polar Axes or Polar Theta-Coordinate Method



34.5.5.2. Optimizations

Additional optimizations can be applied to improve the quality of the mesh partitions. The heuristic of bi-secting perpendicular to the direction of longest domain extent is not always the best choice for creating the smallest interface boundary. A pre-testing operation, (For details, see *Pretesting* (p. 1621)) can be applied to automatically choose the best direction before partitioning. In addition, the following iterative optimization schemes exist:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smooth

attempts to minimize the number of partition interfaces by swapping cells between partitions. The scheme traverses the partition boundary and gives cells to the neighboring partition if the interface boundary surface area is decreased. (See *Figure 34.20* (p. 1621).)

Merge

attempts to eliminate orphan clusters from each partition. An orphan cluster is a group of cells with the common feature that each cell within the group has at least one face which coincides with an interface boundary. (See *Figure 34.21* (p. 1621).) Orphan clusters can degrade multigrid performance and lead to large communication costs.

Figure 34.20 The Smooth Optimization Scheme

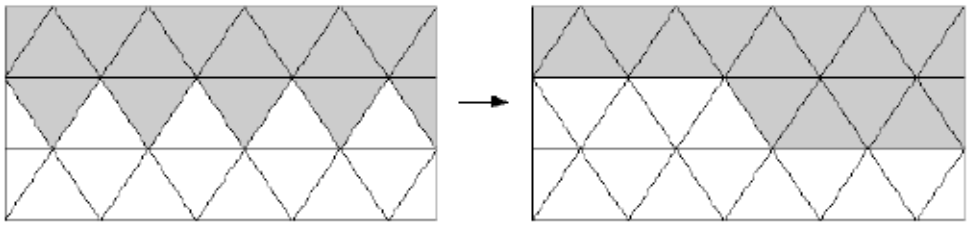
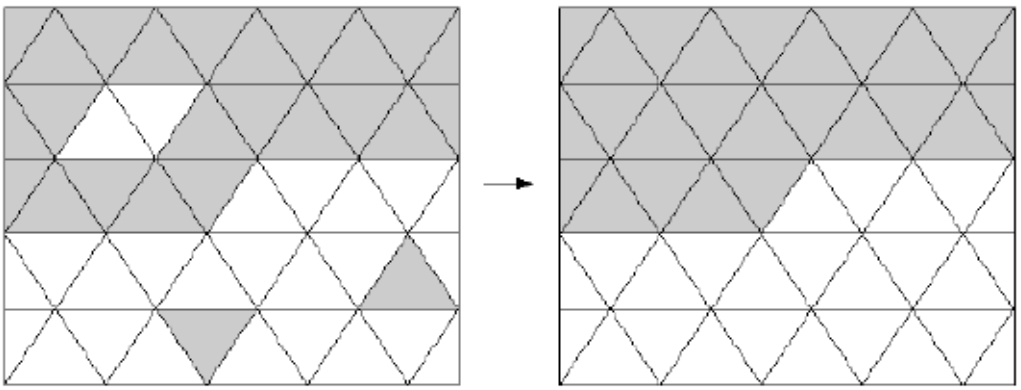


Figure 34.21 The Merge Optimization Scheme



In general, the Smooth and Merge schemes are relatively inexpensive optimization tools.

34.5.5.3. Pretesting

If you choose the **Principal Axes** or **Cartesian Axes** method, you can improve the bisection by testing different directions before performing the actual bisection. If you choose not to use pretesting (the default), ANSYS FLUENT will perform the bisection perpendicular to the direction of longest domain extent.

If pretesting is enabled, it will occur automatically when you click the **Partition** button in the *Partitioning and Load Balancing Dialog Box* (p. 2161), or when you read in the mesh if you are using automatic partitioning. The bisection algorithm will test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that using pretesting will increase the time required for partitioning. For 2D problems partitioning will take 3 times longer than without pretesting, and for 3D problems it will take 4 times longer.

34.5.5.4. Using the Partition Filter

As noted above, you can use the METIS partitioning method through a filter in addition to within the **Auto Partition Mesh** and **Partitioning and Load Balancing** dialog boxes. To perform METIS partitioning on an unpartitioned mesh, use the **File/Import/Partition/Metis...** menu item.

File → **Import** → **Partition** → **Metis...**

ANSYS FLUENT will use the METIS partitioner to partition the mesh, and then read the partitioned mesh into the solver. The number of partitions will be equal to the number of processes. You can then proceed with the model definition and solution.

Important

Direct import to the parallel solver through the partition filter requires that the host machine has enough memory to run the filter for the specified mesh. If not, you will need to run the filter on a machine that does have enough memory. You can either start the parallel solver on the machine with enough memory and repeat the process described above, or run the filter manually on the new machine and then read the partitioned mesh into the parallel solver on the host machine.

To manually partition a mesh using the partition filter, enter the following command:

`utility partition` *input_filename partition_count output_filename*

where *input_filename* is the filename for the mesh to be partitioned, *partition_count* is the number of partitions desired, and *output_filename* is the filename for the partitioned mesh. You can then read the partitioned mesh into the solver (using the standard **File/Read/Case...** menu item) and proceed with the model definition and solution.

When the **File/Import/Partition/Metis...** menu item is used to import an unpartitioned mesh into the parallel solver, the METIS partitioner partitions the entire mesh. You may also partition each cell zone individually, using the **File/Import/Partition/Metis Zone...** menu item.

File → **Import** → **Partition** → **Metis Zone...**

This method can be useful for balancing the work load. For example, if a case has a fluid zone and a solid zone, the computation in the fluid zone is more expensive than in the solid zone, so partitioning each zone individually will result in a more balanced work load.

34.5.6. Checking the Partitions

After partitioning a mesh, you should check the partition information and examine the partitions graphically.

34.5.6.1. Interpreting Partition Statistics

You can request a report to be displayed after partitioning (either automatic or manual) is completed. In the parallel solver, click the **Print Active Partitions** or **Print Stored Partitions** button in the **Partitioning and Load Balancing** dialog box. In the serial solver, click the **Print Partitions** button.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS FLUENT distinguishes between two cell partition schemes within a parallel problem: the active cell partition and the stored cell partition. Initially, both are set to the cell partition that was established upon reading the case file. If you re-partition the mesh using the **Partitioning and Load Balancing** dialog box, the new partition will be referred to as the stored cell partition. To make it the active cell partition, you need to click the **Use Stored Partitions** button in the **Partitioning and Load Balancing** dialog box. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. This distinction is made mainly to allow you to partition a case on one machine or network of machines and solve it on a different one. Thanks to the two separate partitioning schemes, you could use the parallel solver with a certain number of compute nodes to subdivide a mesh into an arbitrary different number of partitions, suitable for a different parallel machine, save the case file, and then load it into the designated machine.

When you click **Print Partitions** in the serial solver, you will obtain information about the stored partition.

The output generated by the partitioning process includes information about the recursive subdivision and iterative optimization processes. This is followed by information about the final partitioned mesh, including the partition ID, number of cells, number of faces, number of interface faces, ratio of interface faces to faces for each partition, number of neighboring partitions, and cell, face, interface, neighbor, mean cell, face ratio, and global face ratio variations. Global face ratio variations are the minimum and maximum values of the respective quantities in the present partitions. For example, in the sample output below, partitions 0 and 3 have the minimum number of interface faces (10), and partitions 1 and 2 have the maximum number of interface faces (19); hence the variation is 10–19.

Your aim is to achieve small values of `Interface ratio variation` and `Global interface ratio` while maintaining a balanced load (`Cell variation`).

```
 >> Partitions:
P    Cells    I-Cells     Cell Ratio     Faces    I-Faces     Face Ratio   Neighbors
0      134       10            0.075       217         10          0.046          1
1      137       19            0.139       222         19          0.086          2
2      134       19            0.142       218         19          0.087          2
3      137       10            0.073       223         10          0.045          1
   ------
   Partition count             = 4
   Cell variation              = (134 - 137)
   Mean cell variation         = (-1.1% - 1.1%)
   Intercell variation         = (10 - 19)
   Intercell ratio variation   = (7.3% - 14.2%)
   Global intercell ratio      =  10.7%
   Face variation              = (217 - 223)
   Interface variation         = (10 - 19)
   Interface ratio variation   = (4.5% - 8.7%)
   Global interface ratio      = 3.4%
   Neighbor variation          = (1 - 2)

 Computing connected regions; type ^C to interrupt.
   Connected region count      = 4
```

Note that partition IDs correspond directly to compute node IDs when a case file is read into the parallel solver. When the number of partitions in a case file is larger than the number of compute nodes, but is evenly divisible by the number of compute nodes, then the distribution is such that partitions with IDs 0 to $(M-1)$ are mapped onto compute node 0, partitions with IDs M to $(2M-1)$ onto compute node 1, etc., where M is equal to the ratio of the number of partitions to the number of compute nodes.

34.5.6.2. Examining Partitions Graphically

To further aid interpretation of the partition information, you can draw contours of the mesh partitions, as illustrated in

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graphics and Animations → **Contours** → **Set Up...**

To display the active cell partition or the stored cell partition (which were described above), select **Active Cell Partition** or **Stored Cell Partition** in the **Cell Info...** category of the **Contours Of** drop-down list, and turn off the display of **Node Values**. (For details, see *Displaying Contours and Profiles* (p. 1386) for information about displaying contours.)

Important

If you have not already done so in the setup of your problem, you will need to perform a solution initialization in order to use the **Contours** dialog box.

34.5.7. Load Distribution

If the speeds of the processors that will be used for a parallel calculation differ significantly, you can specify a load distribution for partitioning, using the `load-distribution` text command.

`parallel` → `partition` → `set` → `load-distribution`

For example, if you will be solving on three compute nodes, and one machine is twice as fast as the other two, then you may want to assign twice as many cells to the first machine as to the others (i.e., a load vector of (2 1 1)). During subsequent mesh partitioning, partition 0 will end up with twice as many cells as partitions 1 and 2.

For this example, you need to start up ANSYS FLUENT such that compute node 0 is the fast machine, since partition 0, with twice as many cells as the others, will be mapped onto compute node 0. Alternatively, in this situation, you could enable the load balancing feature (described in *Load Balancing* (p. 1614)) to have ANSYS FLUENT automatically attempt to discern any difference in load among the compute nodes.

34.5.8. Troubleshooting

When running a calculation using parallel ANSYS FLUENT, you may encounter a warning message in the console that reports problems related to the partitioning. The following is an example of such a warning:

```
#AMG# Warning: The global matrix size (1273286) is too large, and may
adversely affect the parallel performance.  See the ANSYS FLUENT User's
Guide for information on troubleshooting partitioning issues.
```

The following are possible reasons why the global matrix size is so large, along with recommendations for reducing it:

- The presence of solid zones may cause a partition to have a very small amount of fluid cells. For such cases, it is recommended that you partition the mesh with the **Across Zones** option disabled in the **Partitioning and Load Balancing** dialog box, and then click the **Use Stored Partitions** button.
- A partition may have a small number of cells if you have set up a load distribution for partitioning. Such settings should be disabled, by using the `load-distribution` text command (described in *Load Distribution* (p. 1624)) and entering a value of `1` for each of the previously defined partitions.
- Some model settings (e.g., shell conduction) can encapsulate some cells, which may cause difficulties with the coarsening process. To remedy this situation, you can either try a different partitioning method, or you can enable the global coarsening checking criteria with the following rpvar setting:

  ```
  (rpsetvar 'amg/parallel/global-check-coarsening? #t)
  ```

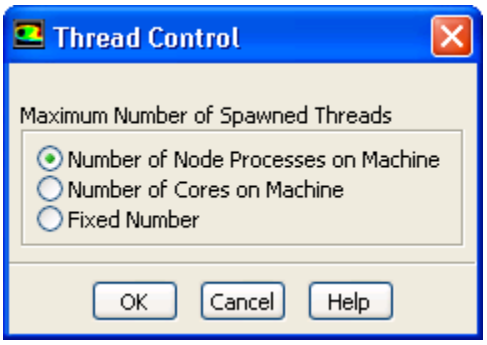

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6. Controlling the Threads

You can control the maximum number of threads on each machine by using the **Thread Control** dialog box (*Figure 34.23* (p. 1626)).

Parallel → **Thread Control...**

Figure 34.22 The Thread Control Dialog Box



You have the following options when using the **Thread Control** dialog box:

- **Number of Node Processes on Machine**

  This is the default option. When this option is chosen, the maximum number of threads on each machine is equal to the number of ANSYS FLUENT node processes on each machine.

- **Number of Cores on Machine**

  When this option is chosen, the maximum number of threads on each machine is equal to the number of cores on the machine. ANSYS FLUENT obtains the number of cores from the OS. This may be applicable when the multi-threaded part of the calculation is dominating the computation time, and the continuous phase calculation is relatively small, and you want to take full advantage of the computation resources. For example, if you have a very small case with regard to the number of cells, but a large number of particles to be tracked, you may want to spawn one ANSYS FLUENT node process on each machine, but use the maximum number of cores in order to get a good overall performance.

- **Fixed Number**

  When this option is chosen, you may input the maximum number of threads that can be spawned on each machine in the number-entry box below **Fixed Number**. This may only be applicable when you want to have fine control of the number of threads on each machine; it is not recommended in general.

34.7. Checking Network Connectivity

For any compute node, you can print network connectivity information that includes the hostname, architecture, process ID, and ID of the selected compute node and all machines connected to it. The ID of the selected compute node is marked with an asterisk.

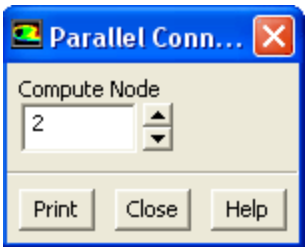
The ID for the ANSYS FLUENT host process is always `host`. The compute nodes are numbered sequentially starting from `node-0`. All compute nodes are completely connected. In addition, compute node 0 is connected to the host process.

To obtain connectivity information for a compute node, you can use the *Parallel Connectivity Dialog Box* (p. 2171) (*Figure 34.23* (p. 1626)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel → **Network** → **Show Connectivity...**

Figure 34.23 The Parallel Connectivity Dialog Box



Indicate the compute node ID for which connectivity information is desired in the **Compute Node** field, and then click the **Print** button. Sample output for compute node 0 is shown below:

```
-----------------------------------------------------------------------------
ID      Comm.    Hostname    O.S.        PID      Mach ID   HW ID    Name
-----------------------------------------------------------------------------
host    hp       balin      Linux-32     17272    0         7        Fluent Host
n3      hp       balin      Linux-32     17307    1         10       Fluent Node
n2      hp       filio      Linux-32     17306    0         -1       Fluent Node
n1      hp       bofur      Linux-32     17305    0         1        Fluent Node
n0*     hp       balin      Linux-32     17273    2         11       Fluent Node
```

`O.S` is the architecture, `Comm.` is the communication library (i.e., MPI type), `PID` is the process ID number, `Mach ID` is the compute node ID, and `HW ID` is an identifier specific to the interconnect used.

34.8. Checking and Improving Parallel Performance

To determine how well the parallel solver is working, you can measure computation and communication times, and the overall parallel efficiency, using the performance meter. You can also control the amount of communication between compute nodes in order to optimize the parallel solver, and take advantage of the automatic load balancing feature of ANSYS FLUENT.

Information about checking and improving parallel performance is provided in the following sections:

34.8.1. Checking Parallel Performance
34.8.2. Optimizing the Parallel Solver

34.8.1. Checking Parallel Performance

The performance meter allows you to report the wall clock time elapsed during a computation, as well as message-passing statistics. Since the performance meter is always activated, you can access the statistics by displaying them after the computation is completed. To view the current statistics, use the **Parallel/Timer/Usage** menu item.

Parallel → **Timer** → **Usage**

Performance statistics will be displayed in the console.

To clear the performance meter so that you can eliminate past statistics from the future report, use the **Parallel/Timer/Reset** menu item.

Parallel → **Timer** → **Reset**

The following example demonstrates how the current parallel statistics are displayed in the console:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Performance Timer for 1 iterations on 4 compute nodes
Average wall-clock time per iteration:            4.901 sec
Global reductions per iteration:                    408 ops
Global reductions time per iteration:             0.000 sec (0.0%)
Message count per iteration:                        801 messages
Data transfer per iteration:                      9.585 MB
LE solves per iteration:                            12 solves
LE wall-clock time per iteration:                 2.445 sec (49.9%)
LE global solves per iteration:                     27 solves
LE global wall-clock time per iteration:          0.246 sec (5.0%)
AMG cycles per iteration:                           64 cycles
Relaxation sweeps per iteration:                  4160 sweeps
Relaxation exchanges per iteration:                 920 exchanges

Total wall-clock time:                            4.901 sec
Total CPU time:                                  17.030 sec
```

A description of the parallel statistics is as follows:

- `Average wall-clock time per iteration` describes the average real (wall clock) time per iteration.
- `Global reductions per iteration` describes the number of global reduction operations (such as variable summations over all processes). This requires communication among all processes.

  A global reduction is a collective operation over all processes for the given job that reduces a vector quantity (the length given by the number of processes or nodes) to a scalar quantity (e.g., taking the sum or maximum of a particular quantity). The number of global reductions cannot be calculated from any other readily known quantities. The number is generally dependent on the algorithm being used and the problem being solved.
- `Global reductions time per iteration` describes the time per iteration for the global reduction operations.
- `Message count per iteration` describes the number of messages sent between all processes per iteration. This is important with regard to communication latency, especially on high-latency interconnnects.

  A message is defined as a single point-to-point, send-and-receive operation between any two processes. This excludes global, collective operations such as global reductions. In terms of domain decomposition, a message is passed from the process governing one subdomain to a process governing another (usually adjacent) subdomain.

  The message count per iteration is usually dependent on the algorithm being used and the problem being solved. The message count and the number of messages that are reported are totals for all processors.

  The message count provides some insight into the impact of communication latency on parallel performance. A higher message count indicates that the parallel performance may be more adversely affected if a high-latency interconnect is being used. Ethernet has a higher latency than Myrinet or Infiniband. Thus, a high message count will more adversely affect performance with Ethernet than with Infiniband.

  To check the latency of the overall cluster interconnect, refer to *Checking Latency and Bandwidth* (p. 1628).
- `Data transfer per iteration` describes the amount of data communicated between processors per iteration. This is important with respect to interconnect bandwidth.

  Data transfer per iteration is usually dependent on the algorithm being used and the problem being solved. This number generally increases with increases in problem size, number of partitions, and physics complexity.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The data transfer per iteration may provide some insight into the impact of communication bandwidth (speed) on parallel performance. The precise impact is often difficult to quantify because it is dependent on many things including: ratio of data transfer to calculations, and ratio of communication bandwidth to CPU speed. The unit of data transfer is a byte.

To check the bandwidth of the overall cluster interconnect, refer to *Checking Latency and Bandwidth* (p. 1628).

- `LE solves per iteration`describes the number of linear systems being solved per iteration. This number is dependent on the physics (non-reacting versus reacting flow) and the algorithms (pressure-based versus density-based solver), but is independent of mesh size. For the pressure-based solver, this is usually the number of transport equations being solved (mass, momentum, energy, etc.).
- `LE wall-clock time per iteration` describes the time (wall-clock) spent doing linear equation solvers (i.e., multigrid).
- `LE global solves per iteration` describes the number of solutions on the coarse level of the AMG solver where the entire linear system has been pushed to a single processor (`n0`). The system is pushed to a single processor to reduce the computation time during the solution on that level. Scaling generally is not adversely affected because the number of unknowns is small on the coarser levels.
- `LE global wall-clock time per iteration` describes the time (wall-clock) per iteration for the linear equation global solutions.
- `AMG cycles per iteration` describes the average number of multigrid cycles (V, W, flexible, etc.) per iteration.
- `Relaxation sweeps per iteration` describes the number of relaxation sweeps (or iterative solutions) on all levels for all equations per iteration. A relaxation sweep is usually one iteration of Gauss-Siedel or ILU.
- `Relaxation exchanges per iteration` describes the number of solution communications between processors during the relaxation process in AMG. This number may be less than the number of sweeps because of shifting the linear system on coarser levels to a single node/process.
- `Time-step updates per iteration` describes the number of sub-iterations on the time step per iteration.
- `Time-step wall-clock time per iteration` describes the time per sub-iteration.
- `Total wall-clock time` describes the total wall-clock time.
- `Total CPU time` describes the total CPU time used by all processes. This does not include any wait time for load imbalances or for communications (other than packing and unpacking local buffers).

The most relevant quantity is the `Total wall clock time`. This quantity can be used to gauge the parallel performance (speedup and efficiency) by comparing this quantity to that from the serial analysis (the command line should contain `-t1` in order to obtain the statistics from a serial analysis). In lieu of a serial analysis, an approximation of parallel speedup may be found in the ratio of `Total CPU time` to `Total wall clock time`.

34.8.1.1. Checking Latency and Bandwidth

You can check the latency and bandwidth of the overall cluster interconnect, to help identify any issues affecting ANSYS FLUENT scalability, by using the **Parallel/Network/Show Latency...** and **Parallel/Network/Show Bandwidth...** menu items.

Parallel → **Network** → **Show Latency...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depending on the number of machines and processors being used, a table containing information about the communication speed for each node will be displayed in the console. The table will also summarize the minimum and maximum latency between two nodes.

Consider the following example when checking for latency:

```
Latency (usec) with 1000 samples [1.83128 sec]
 ----------------------------------------
 ID       n0     n1     n2    n3    n4    n5
 ----------------------------------------
 n0            48.0   48.2  48.2  48.3   *50
 n1    48.0           48.2  48.3  48.3   *48
 n2    48.2    48.2         48.8  49.1   *53
 n3    48.2    48.3    *49        48.6  48.5
 n4    48.3    48.3   49.1  48.6         *50
 n5    49.7    48.5    *53  48.5  49.7

 ----------------------------------------
 Min: 47.9956 [n0<-->n1]
 Max: 52.6836 [n5<-->n2]
 ----------------------------------------
```

Important

In the above table, (*) is the maximum value in that row. The smaller the latency, the better.

Six processors (n0 to n5) are spawned. The latency between n0 and n1 is 48.0 μs. Similarly, the latency between n1 and n2 is 48.2 μs. The minimum latency occurs between n0 and n1 and the maximum latency occurs between n2 and n5, as noted in the table. Checking the latency is particularly useful when you are not seeing expected speedup on a cluster.

Parallel $\rightarrow$ Network $\rightarrow$ **Show Bandwidth...**

In addition to checking for latency, you can check your bandwidth. A table containing information about the amount of data communicated within one second between two nodes is displayed in the console. The table will also summarize the minimum and maximum bandwidth between two nodes.

Consider the following example when checking for bandwidth:

```
Bandwidth (MB/s) with 5 messages of size 4MB [4.36388 sec]
 ----------------------------------------
 ID      n0     n1     n2     n3     n4     n5
 ----------------------------------------
 n0          111.8    *55  111.8   97.5  101.3
 n1   111.8          69.2   98.7  111.7    *51
 n2    54.7   69.2          72.9  104.8    *45
 n3   111.8   98.7   72.9          64.0    *45
 n4    97.6  111.7  104.8    *64          76.9
 n5   101.2   50.9   45.5    *45    76.9

 ----------------------------------------
 Min: 45.1039 [n5<-->n3]
 Max: 111.847 [n0<-->n3]
 ----------------------------------------
```

Important

In the above table, (*) is the minimum value in that row. The larger the bandwidth, the better.

The bandwidth between n0 and n1 is 111.8 MB/s. Similarly, the bandwidth between n1 and n2 is 69.2 MB/s. The minimum amount of bandwidth occurs between n3 and n5 and the maximum occurs between n0 and

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

n3, as noted in the table. Checking the bandwidth is particularly useful when you cannot see good scalability with relatively large cases.

34.8.2. Optimizing the Parallel Solver

34.8.2.1. Increasing the Report Interval

In ANSYS FLUENT, you can reduce communication and improve parallel performance by increasing the report interval for residual printing/plotting or other solution monitoring reports. You can modify the value for **Reporting Interval** in the *Run Calculation Task Page* (p. 1955).

Run Calculation → **Calculate...**

Important

Note that you will be unable to interrupt iterations until the end of each report interval.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

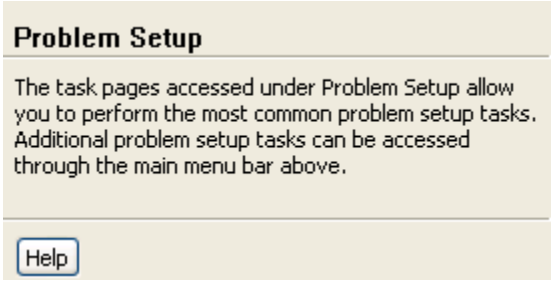
Chapter 35: Task Page Reference Guide

This reference guide provides information about the task pages in FLUENT.

35.1. Problem Setup Task Page
35.2. General Task Page
35.3. Models Task Page
35.4. Materials Task Page
35.5. Phases Task Page
35.6. Cell Zone Conditions Task Page
35.7. Boundary Conditions Task Page
35.8. Mesh Interfaces Task Page
35.9. Dynamic Mesh Task Page
35.10. Reference Values Task Page
35.11. Solution Task Page
35.12. Solution Methods Task Page
35.13. Solution Controls Task Page
35.14. Monitors Task Page
35.15. Solution Initialization Task Page
35.16. Calculation Activities Task Page
35.17. Run Calculation Task Page
35.18. Results Task Page
35.19. Graphics and Animations Task Page
35.20. Plots Task Page
35.21. Reports Task Page

35.1. Problem Setup Task Page

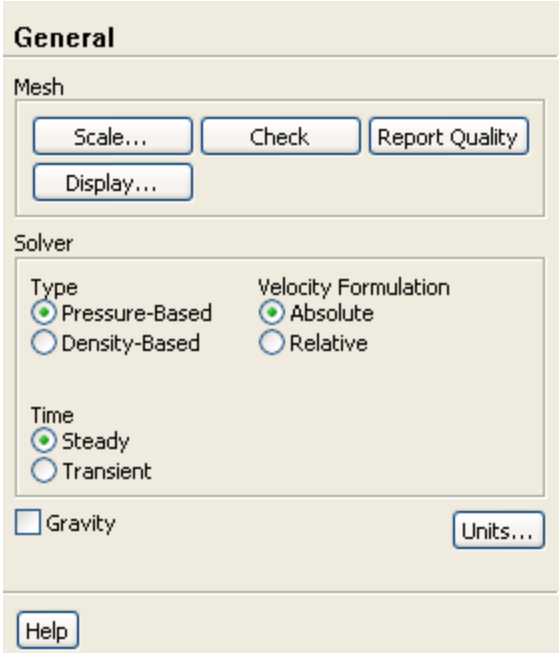
The **Problem Setup** task page introduces you to the main tasks involved in setting up your CFD simulation using ANSYS FLUENT.



35.2. General Task Page

The **General** task page allows you to set various generic problem settings, such as those related to the mesh or the solver.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mesh

contains controls relating to mesh settings.

Scale...

opens the *Scale Mesh Dialog Box* (p. 1634).

Check

verifies the validity of the mesh. The check provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases. See *Checking the Mesh* (p. 172) for details.

It is recommended that you check the mesh right after reading it into the solver, in order to detect any mesh trouble before you get started with the problem setup.

Report Quality

displays various quantities related to the quality of the mesh in the console, such as `Minimum Orthogonal Quality` and `Maximum Aspect Ratio`. See *Mesh Quality* (p. 144) for details.

Display...

opens the *Mesh Display Dialog Box* (p. 1635).

Solver

contains controls relating to solver settings.

Type

contains the solution methods available for computing a solution for your model. See *Using the Solver* (p. 1207) for details.

Pressure-Based

enables the pressure-based Navier-Stokes solution algorithm (the default).

Density-Based

enables the density-based Navier-Stokes coupled solution algorithm.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocity Formulation

specifies the velocity formulation to be used in the calculation. See *Choosing the Relative or Absolute Velocity Formulation* (p. 518) for details.

Absolute

enables the use of the absolute velocity formulation. This is the default setting.

Relative

enables the use of the relative velocity formulation. This option is available only with the **Pressure-Based** solver.

Time

contains options related to time dependence.

Steady

specifies that a steady flow is being solved.

Transient

enables a time-dependent solution. See *Performing Time-Dependent Calculations* (p. 1255) for details.

2D Space

contains options available only when solving two-dimensional problems.

Planar

indicates that the problem is two-dimensional. (This option is available only when you start the 2D version of the solver.)

Axisymmetric

indicates that the domain is axisymmetric about the x axis. When **Axisymmetric** is enabled, the 2D axisymmetric form of the governing equations is solved instead of the 2D Cartesian form. (This option is available only when you start the 2D version of the solver.) Be sure to change the zone type of the axis of rotation to **axis**, using the *Boundary Conditions Task Page* (p. 1812), as described in *Changing Cell and Boundary Zone Types* (p. 207).

Axisymmetric Swirl

specifies that the swirl component (circumferential component) of velocity is to be included in your axisymmetric model. You should select this option if you are solving swirling flow in an axisymmetric geometry (see *Swirling and Rotating Flows* (p. 497) for more information).

Gravity

enables the specification of gravity.

Gravitational Acceleration

sets the x, y, and z components of the gravitational acceleration vector. (The z component is available only in 3D solvers.) See *Natural Convection and Buoyancy-Driven Flows* (p. 697) for details about buoyancy-driven flows. This option appears only when **Gravity** is enabled.

This quantity can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Units...

opens the *Set Units Dialog Box* (p. 1637).

For additional information, please see the following sections:

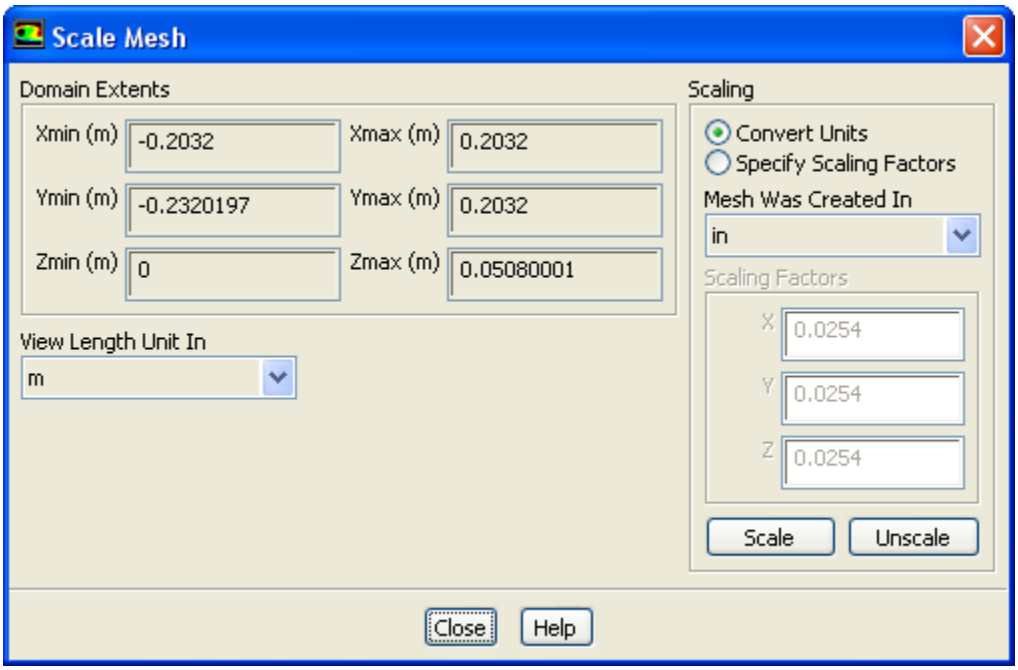
35.2.1. Scale Mesh Dialog Box
35.2.2. Mesh Display Dialog Box
35.2.3. Set Units Dialog Box
35.2.4. Define Unit Dialog Box

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.2.5. Mesh Colors Dialog Box

35.2.1. Scale Mesh Dialog Box

The **Scale Mesh** dialog box allows you to convert the mesh from various units of measurement to SI or to apply custom scale factors to the individual coordinates of the mesh. See *Scaling the Mesh* (p. 200) for details.



Controls

Domain Extents

displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Xmin, Ymin, Zmin

shows the minimum values of Cartesian coordinates in the mesh.

Xmax, Ymax, Zmax

shows the maximum values of Cartesian coordinates in the mesh.

Scaling

contains controls for converting units and setting the scale factors automatically.

Convert Units

allows you to use the conversion factors provided by ANSYS FLUENT. Then indicate the units used when creating the mesh by selecting the appropriate abbreviation for meters, centimeters, millimeters, inches, or feet from the **Mesh Was Created In** drop-down list. The **Scaling Factors** will automatically be set to the correct values (e.g., 0.0254 meters/inch).

Specify Scaling Factors

allows you to manually specify a scale factor in each of the Cartesian coordinate directions.

Mesh Was Created In

contains a list of common units of length. The **Scaling Factors** will automatically be set based on your selection. The units include common SI and British units such as centimeters (cm), millimeters (mm), inches (in), and feet (ft).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scaling Factors

contains the factors applied to the mesh in each of the Cartesian coordinate directions. You can enter values manually, or use the **Mesh Was Created In** list to set scale factors automatically.

X

is the scale factor in the x direction.

Y

is the scale factor in the y direction.

Z

is the scale factor in the z direction (appears only in 3D).

Scale

multiplies each of the node coordinates by the specified scale factors.

Unscale

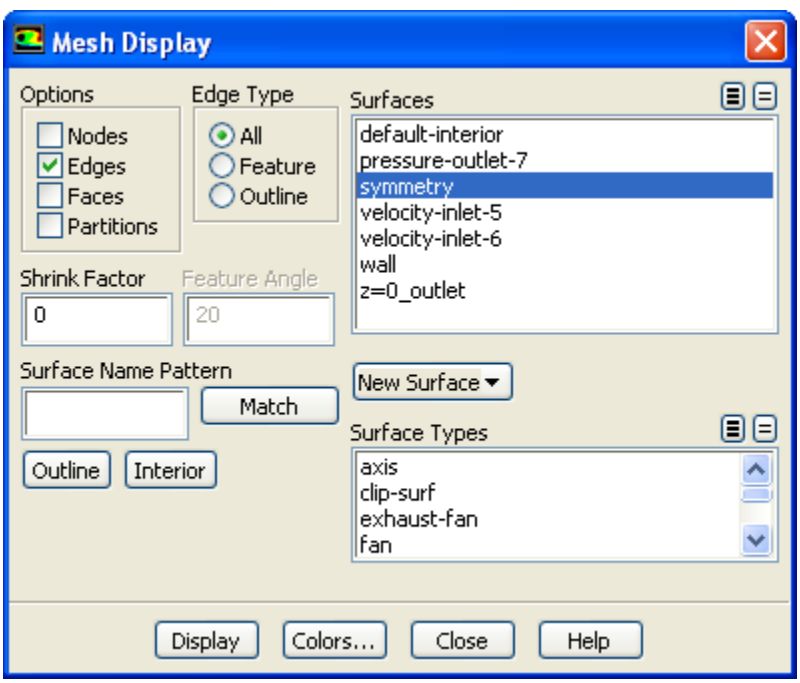
divides each of the node coordinates by the specified scale factors.

View Length Unit In

contains a list of common units of length. The **Domain Extents** will automatically be set based on your selection. The units include common SI and British units such as centimeters (cm), millimeters (mm), inches (in), and feet (ft). The units of length in the *Set Units Dialog Box* (p. 1637) will change each time you change your selection in the **View Length Unit In** drop-down list.

35.2.2. Mesh Display Dialog Box

The **Mesh Display** dialog box controls the display of zone, surface, and partition boundary meshes. See *Displaying the Mesh* (p. 1379) for details about the items below.



Controls

Options

contains the rendering options described below. To see the effects of your selection you must click the **Display** button.

Nodes
enables the display of nodes on the selected surfaces.

Edges
enables the display of mesh edges on the selected surfaces.

Faces
enables the display of mesh faces (filled meshes) on the selected surfaces.

Partitions
enables the display of partition boundaries.

Shrink Factor
specifies the amount to shrink faces and cells. See *Shrinking Faces and Cells in the Display* (p. 1385) for details.

Edge Type
controls the display of edges. (These items will not appear if the **Edges** option is turned off.)

All
enables the display of all mesh edges.

Feature
enables feature lines in an outline display. See *Adding Features to an Outline Display* (p. 1384) for details.

Outline
enables the display of the mesh outline.

Feature Angle
controls the amount of detail added to a feature outline display. See *Adding Features to an Outline Display* (p. 1384) for details. (This item will be available only if the **Feature** edge type is enabled.)

Surface Name Pattern
specifies the pattern to look for in the names of surfaces. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Surfaces** list with names that match the specified pattern. See *Generating Mesh or Outline Plots* (p. 1381) for information about matching additional characters using * and ?.

Surfaces
contains a list from which you can select the surfaces for which the mesh is to be drawn.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the *Point Surface Dialog Box* (p. 1927).

Line/Rake
opens the *Line/Rake Surface Dialog Box* (p. 1928).

Plane
opens the *Plane Surface Dialog Box* (p. 1929).

Quadric
opens the *Quadric Surface Dialog Box* (p. 1931).

Iso-Surface
opens the *Iso-Surface Dialog Box* (p. 1933).

Iso-Clip
opens the *Iso-Clip Dialog Box* (p. 1935).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Types

contains a selectable list of types of surfaces. If you select (or deselect) an item in this list, all surfaces of that type will be selected (or deselected) automatically in the **Surfaces** list.

Outline

selects all "outline" boundaries in the **Surfaces** list. If all outline boundaries are already selected, it deselects them.

Interior

selects all "interior" surfaces in the **Surfaces** list. If all interior surfaces are already selected, it deselects them.

Display

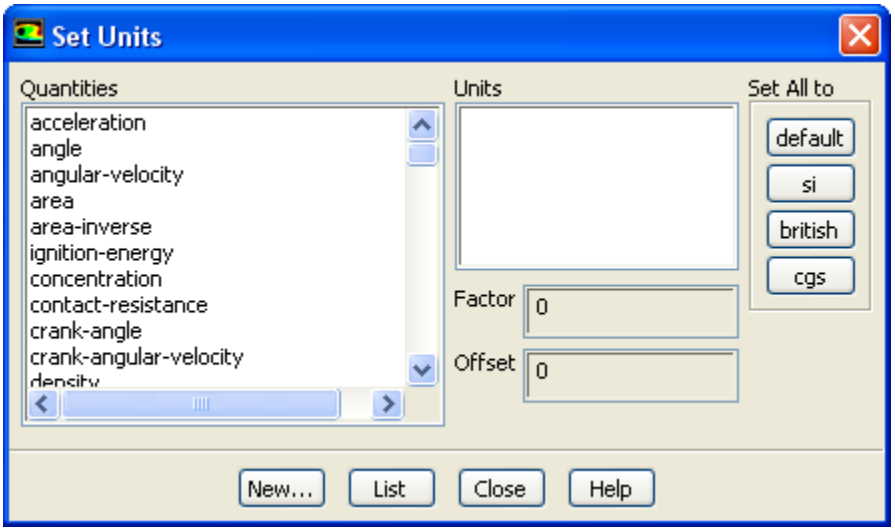
displays the defined mesh plot.

Colors...

opens the *Mesh Colors Dialog Box* (p. 1638).

35.2.3. Set Units Dialog Box

The **Set Units** dialog box allows you to set the units system for any quantity used in ANSYS FLUENT. All quantities may be set to a standard system, such as SI or British, or the units of individual quantities may be set. See *Unit Systems* (p. 125) for details about the items below.



Controls

Quantities

displays the list of all quantities used by ANSYS FLUENT for input and output.

Units

lists the units appropriate for the currently selected quantity. Selecting an item in the **Units** list causes that unit to be used for the currently selected quantity.

Factor

displays the conversion factor from the currently selected units to SI.

Offset

displays the conversion offset from the currently selected units to SI.

Set All to

contains standard sets of units that are applied to all quantities when selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default
is similar to **si**, but uses degrees instead of radians for angles.

si
selects the System International (SI) standard for all units.

british
selects the English Engineering standard for all units.

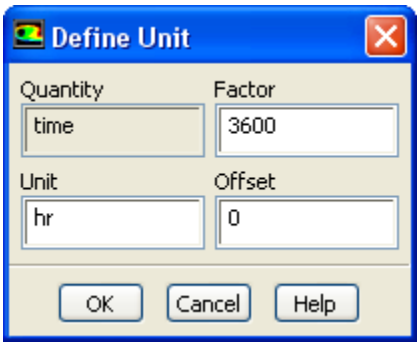
cgs
selects the CGS (centimeter-gram-second) standard for all units

New...
opens the *Define Unit Dialog Box* (p. 1638), in which you can specify a customized unit for a particular quantity.

List
displays the current units for all quantities in the console.

35.2.4. Define Unit Dialog Box

The **Define Unit** dialog box allows you to customize the units for a particular quantity. It is opened from the *Set Units Dialog Box* (p. 1637). For details about using this dialog box, see *Defining a New Unit* (p. 127).



Controls

Quantity
shows the name of the quantity for which you are defining a new unit. (You cannot edit this field; the quantity is selected in the *Set Units Dialog Box* (p. 1637).)

Unit
sets the name for the new unit.

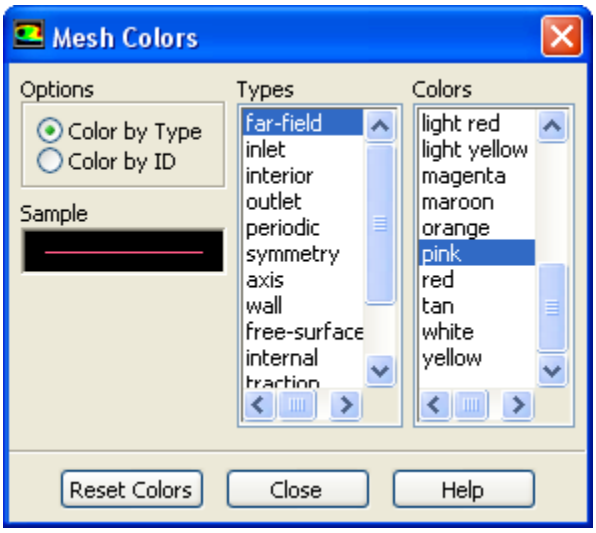
Factor
sets the conversion factor from the currently selected units to SI.

Offset
sets the conversion offset from the currently selected units to SI.

35.2.5. Mesh Colors Dialog Box

The **Mesh Colors** dialog box allows you to control the colors that are used to draw meshes. It is opened from the *Mesh Display Dialog Box* (p. 1635). See *Modifying the Mesh Colors* (p. 1383) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains options to select the method by which to set the colors.

Color by Type
sets the color based on the type of zone.

Color by ID
sets the color by ID.

Types
contains a selectable list of zone types. You can select the zone type for which you want to set the color.

Colors
contains a list from which you can select a color for the selected type.

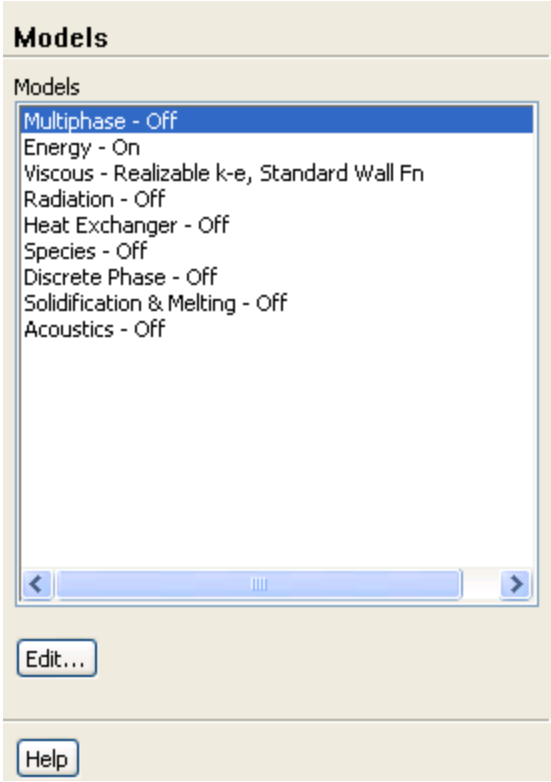
Sample
displays a sample of the currently selected color.

Reset Colors
resets the mesh colors to the default selections.

35.3. Models Task Page

The **Models** task page allows you to set various generic model settings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Models

contains a listing of the various models available in ANSYS FLUENT.

You can double-click an item in the **Models** list to open the corresponding dialog box, or you can select the item in the list and click the **Edit...** button.

Multiphase

- selecting this item and clicking the **Edit...** button opens the *Multiphase Model Dialog Box* (p. 1642).

Energy

- selecting this item and clicking the **Edit...** button opens the *Energy Dialog Box* (p. 1645).

Viscous

- selecting this item and clicking the **Edit...** button opens the *Viscous Model Dialog Box* (p. 1645).

Radiation

- selecting this item and clicking the **Edit...** button opens the *Radiation Model Dialog Box* (p. 1657).

Heat Exchanger

- selecting this item and clicking the **Edit...** button opens the *Heat Exchanger Model Dialog Box* (p. 1666).

Species

- selecting this item and clicking the **Edit...** button opens the *Species Model Dialog Box* (p. 1681).

The following models are made available, depending on your setup of the **Species** dialog box.

- **Spark Ignition** - selecting this item and clicking the **Edit...** button opens the *Spark Ignition Dialog Box* (p. 1700). Note that spark ignition is only available for transient calculations.
- **Autoignition** - selecting this item and clicking the **Edit...** button opens the *Autoignition Model Dialog Box* (p. 1703). Note that autoignition is only available for transient calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Inert** - selecting this item and clicking the **Edit...** button opens the *Inert Dialog Box* (p. 1705). Note that the inert model is only available when the non-premixed or partially premixed model is selected in the **Species Model** dialog box, or when a PDF file is read.
- **NOx** - selecting this item and clicking the **Edit...** button opens the *NOx Model Dialog Box* (p. 1706). Note that the NOx model is not compatible with premixed combustion.
- **SOx** - selecting this item and clicking the **Edit...** button opens the *SOx Model Dialog Box* (p. 1713). Note that the SOx model is not compatible with premixed combustion.
- **Soot** - selecting this item and clicking the **Edit...** button opens the *Soot Model Dialog Box* (p. 1717). Note that none of the soot models are compatible with premixed combustion.
- **Decoupled Detailed Chemistry** - selecting this item and clicking the **Edit...** button opens the *Decoupled Detailed Chemistry Dialog Box* (p. 1722).

Discrete Phase
- selecting this item and clicking the **Edit...** button opens the *Discrete Phase Model Dialog Box* (p. 1723).

Solidification & Melting
- selecting this item and clicking the **Edit...** button opens the *Solidification and Melting Dialog Box* (p. 1730).

Acoustics
- selecting this item and clicking the **Edit...** button opens the *Acoustics Model Dialog Box* (p. 1731).

Edit...
displays the dialog box corresponding to the selected item in the **Models** list.

For additional information, please see the following sections:

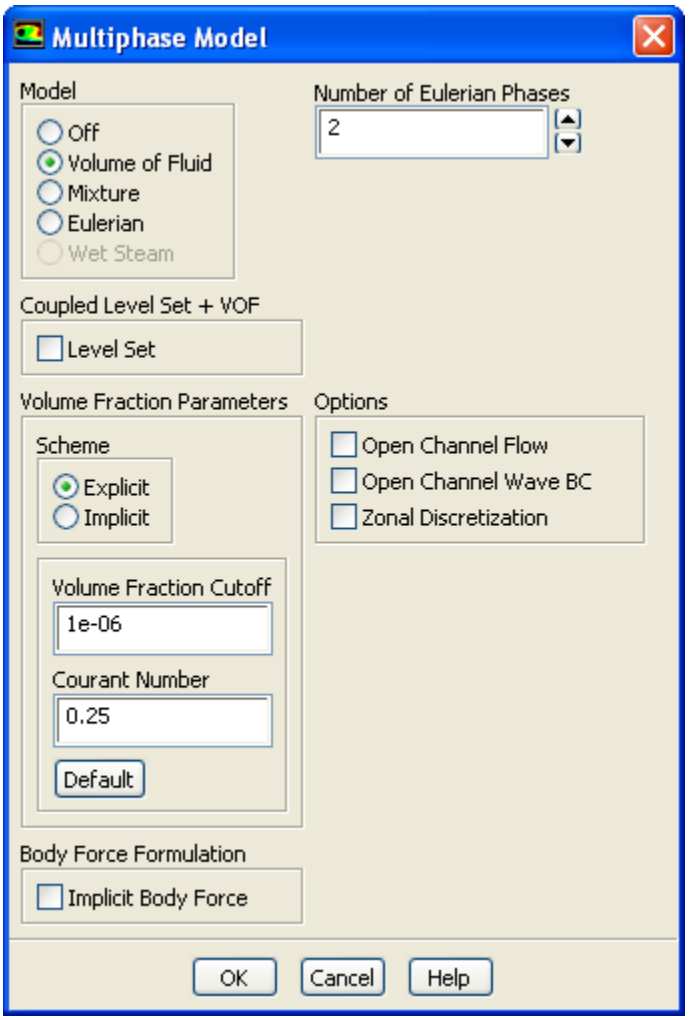
35.3.1. Multiphase Model Dialog Box
35.3.2. Energy Dialog Box
35.3.3. Viscous Model Dialog Box
35.3.4. Radiation Model Dialog Box
35.3.5. View Factors and Clustering Dialog Box
35.3.6. Participating Boundary Zones Dialog Box
35.3.7. Solar Calculator Dialog Box
35.3.8. Heat Exchanger Model Dialog Box
35.3.9. Dual Cell Heat Exchanger Dialog Box
35.3.10. Set Dual Cell Heat Exchanger Dialog Box
35.3.11. Heat Transfer Data Table Dialog Box
35.3.12. NTU Table Dialog Box
35.3.13. Copy From Dialog Box
35.3.14. Ungrouped Macro Heat Exchanger Dialog Box
35.3.15. Velocity Effectiveness Curve Dialog Box
35.3.16. Core Porosity Model Dialog Box
35.3.17. Macro Heat Exchanger Group Dialog Box
35.3.18. Species Model Dialog Box
35.3.19. Coal Calculator Dialog Box
35.3.20. Integration Parameters Dialog Box
35.3.21. Chemkin Mechanism Import Dialog Box
35.3.22. Flamelet 3D Surfaces Dialog Box
35.3.23. Flamelet 2D Curves Dialog Box
35.3.24. Spark Ignition Dialog Box
35.3.25. Set Spark Ignition Dialog Box
35.3.26. Autoignition Model Dialog Box
35.3.27. Inert Dialog Box

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3.28. NOx Model Dialog Box
35.3.29. SOx Model Dialog Box
35.3.30. Soot Model Dialog Box
35.3.31. Decoupled Detailed Chemistry Dialog Box
35.3.32. Discrete Phase Model Dialog Box
35.3.33. Solidification and Melting Dialog Box
35.3.34. Acoustics Model Dialog Box
35.3.35. Acoustic Sources Dialog Box
35.3.36. Acoustic Receivers Dialog Box
35.3.37. Interior Cell Zone Selection Dialog Box

35.3.1. Multiphase Model Dialog Box

The **Multiphase Model** dialog box allows you to set parameters for modeling multiphase flow. See *Enabling the Multiphase Model* (p. 1093) – *Including Cavitation Effects* (p. 1153) for details.



Controls

Model

allows you to select one of four multiphase models.

Off

disables the calculation of multiphase flow.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume of Fluid

enables the VOF model described in Volume of Fluid (VOF) Model Theory in the Theory Guide. See *Setting Up the VOF Model* (p. 1120) for details about using the model. This is available only with the pressure-based solver.

Mixture

enables the mixture model described in Mixture Model Theory in the Theory Guide. See *Setting Up the Mixture Model* (p. 1145) for details about using the model. This is available only with the pressure-based solver.

Eulerian

enables the Eulerian model described in Eulerian Model Theory in the Theory Guide. See *Setting Up the Eulerian Model* (p. 1154) for details about using the model. This is available only with the pressure-based solver.

Wet Steam

enables the wet steam model described in Wet Steam Model Theory in the Theory Guide. See *Setting Up the Wet Steam Model* (p. 1178) for details about using the model. This is available only with the density-based solver.

Number of Eulerian Phases

allows you to specify the number of phases for the multiphase calculation. You can specify up to 20 phases.

Coupled Level Set + VOF

allows you to apply an interface tracking method that couples the level set method with the VOF formulation.

Volume Fraction Parameters

contains parameters related to the VOF and Eulerian model. This section of the dialog box will appear only when **Volume of Fluid** or **Eulerian** is the selected **Model**.

Scheme

allows you to select the desired interface-tracking scheme.

Explicit

enables the Euler explicit scheme, described in The Explicit Scheme in the Theory Guide. See *Choosing a Volume Fraction Formulation* (p. 1095) for more information.

Implicit

enables the implicit scheme, described in The Implicit Scheme in the Theory Guide. See *Choosing a Volume Fraction Formulation* (p. 1095) for more information.

Volume Fraction Cutoff

specifies a cutoff limit for the volume fraction values. The value that you provide is used as the lower cutoff for the volume fraction. All volume fraction values in the domain below this cutoff value are set to zero. The upper cutoff is calculated as (1.0 - lower cutoff). All volume fraction values above the upper cutoff value are set to 1.0. The default value is 1e-6.

Important

The **Volume Fraction Cutoff** value can be specified when using the VOF model, or when using the Eulerian Multiphase model with the **Explicit** scheme.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Courant Number
specifies the maximum Courant number allowed near the free surface. This item will not appear if the **Implicit Scheme** is selected. See *Setting Time-Dependent Parameters for the VOF Model* (p. 1143) for details.

Open Channel Flow
enables the model to study the effects of open channel flow. See Open Channel Flow in the Theory Guide and *Modeling Open Channel Flows* (p. 1121) for details.

Open Channel Wave BC
enables the model to set specific parameters for a particular boundary for open channel wave boundaries. See Backflow Volume Fraction Specification in the Theory Guide and *Modeling Open Channel Wave Boundary Conditions* (p. 1127) for details.

Zonal Discretization
allows you to set the value of the slope limiter. You can select either diffused or sharp interface behavior in different cell zones. This option is available if the **Volume of Fluid** model is used, or if the **Eulerian** model with the **Immiscible Fluid Model** option is enabled.

Mixture Parameters
contains options related to the **Mixture** model.

Slip Velocity
enables/disables the calculation of slip velocities for the secondary phases as described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide. See also *Solving a Homogeneous Multiphase Flow* (p. 1098).

Eulerian Parameters
contains options related to the **Eulerian** model.

Dense Discrete Phase Model
allows you to include the Dense Discrete Phase model (see *Including the Dense Discrete Phase Model* (p. 1167) for details). Enabling this model automatically enables the DPM model.

Immiscible Fluid Model
allows you to include the Immiscible Fluid model (see *Including the Immiscible Fluid Model* (p. 1176) for details). The immiscible fluid model allows the modeling of interface sharpening schemes and free surface flow.

Boiling Model
allows you to include the Boiling model (see *Including the Boiling Model* (p. 1171) for details).

Number of Discrete Phase
allows you to specify the number of discrete phases when the **Dense Discrete Phase Model** option is enabled.

Body Force Formulation
contains an additional option for body force calculations.

Implicit Body Force
enables the implicit body force treatment described in *Including Body Forces* (p. 1099).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

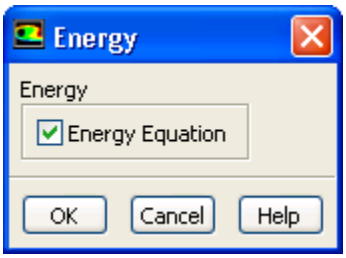
If you want ANSYS FLUENT to solve the volume fraction equation(s) at every iteration within a time step, use the text command:

`define` → `models` → `multiphase` →

and select `vof` as the model. When prompted to `solve vof every iteration?`, enter `yes`.

35.3.2. Energy Dialog Box

The **Energy** dialog box allows you to set parameters related to energy or heat transfer in your model.



Controls

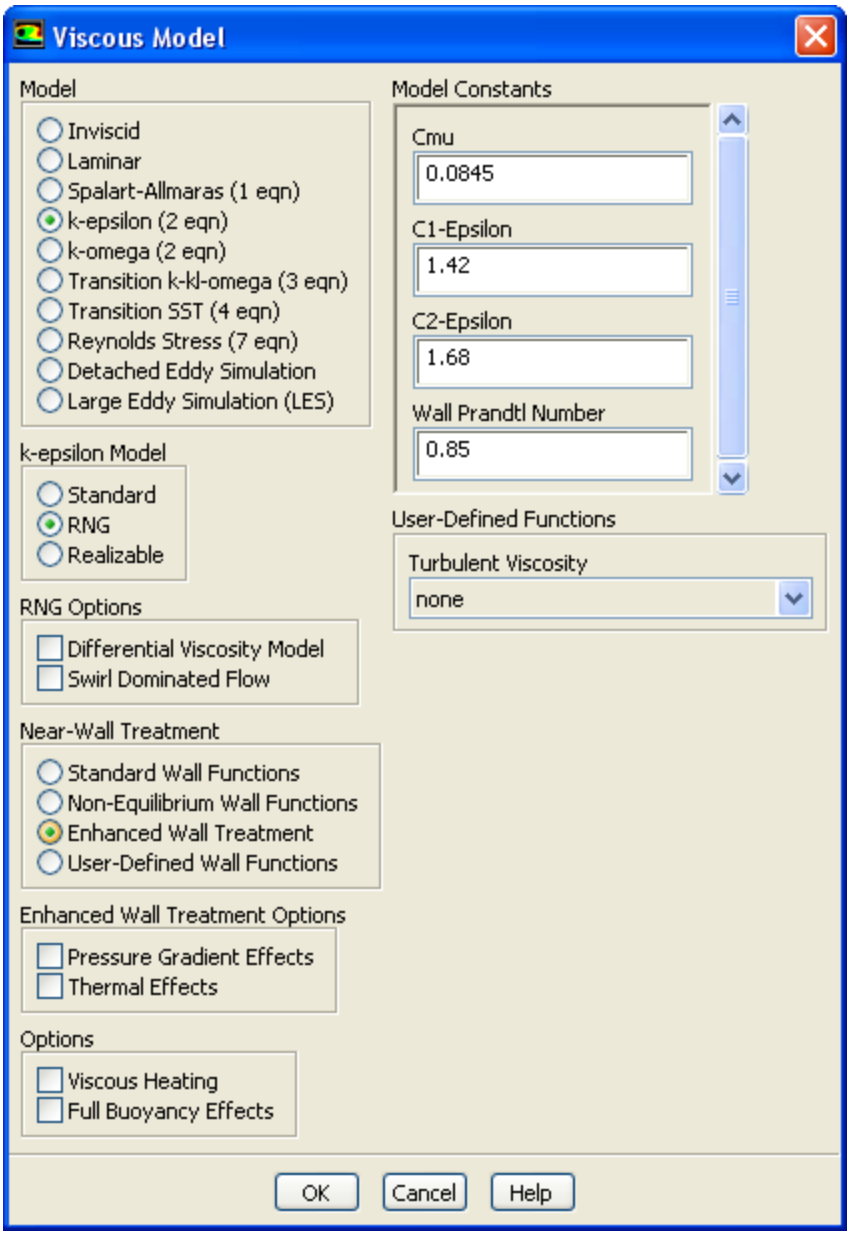
Energy
contains inputs related to the modeling of energy.

Energy Equation
enables/disables the calculation of energy in the model.

35.3.3. Viscous Model Dialog Box

The **Viscous Model** dialog box allows you to set parameters for inviscid, laminar, and turbulent flow. See *Steps in Using a Turbulence Model* (p. 655) for details about using this dialog box to set up a turbulent flow calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains options for specifying the viscous model.

Inviscid

specifies inviscid flow.

Laminar

specifies laminar flow.

Spalart-Allmaras

specifies turbulent flow to be calculated using the Spalart-Allmaras model. (See Spalart-Allmaras Model in the Theory Guide for background about this model. See *Setting Up the Spalart-Allmaras Model* (p. 657) for details about using this model.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

k-epsilon

specifies turbulent flow to be calculated using one of three k- ε models. (See Standard, RNG, and Realizable k- ε Models in the Theory Guide for background about this model. See *Setting Up the k- ε Model* (p. 658) for details about using this model.)

k-omega

specifies turbulent flow to be calculated using one of two k- ω models. (See Standard and SST k- ω Models in the Theory Guide for background about these models. See *Setting Up the k- ω Model* (p. 661) for details about using this model.)

Transition k-kl-omega

specifies turbulent flow to be calculated using the Transition k- kl- ω model. (See k- kl- ω Transition Model in the Theory Guide for background about this model. See *Setting Up the Transition k- kl- ω Model* (p. 662) for details about using this model.)

Transition SST

specifies turbulent flow to be calculated using the Transition SST model. (See Transition SST Model in the Theory Guide for background about this model. See *Setting Up the Transition SST Model* (p. 662) for details about using this model.)

Reynolds Stress

specifies turbulent flow to be calculated using the RSM. (See Reynolds Stress Model (RSM) in the Theory Guide for background about this model. See *Setting Up the Reynolds Stress Model* (p. 663) for details about using this model.)

Scale-Adaptive Simulation

specifies turbulent flow to be unsteady. (See Scale-Adaptive Simulation (SAS) Model in the Theory Guide for background about this model. See *Setting Up the Scale-Adaptive Simulation (SAS) Model* (p. 666) for details about using this model.)

Detached Eddy Simulation

specifies turbulent flow to be calculated using the DES. (See Detached Eddy Simulation (DES) in the Theory Guide for background about this model. See *Setting Up the Detached Eddy Simulation Model* (p. 666) for details about using this model.)

Large Eddy Simulation

(3D only) specifies turbulent flow to be calculated using the LES model. (See Large Eddy Simulation (LES) Model in the Theory Guide for background about this model. See *Setting Up the Large Eddy Simulation Model* (p. 669) for details about using this model.)

Spalart-Allmaras Production

contains options for the Spalart-Allmaras model. This portion of the dialog box will appear only if **Spalart-Allmaras** is selected as the **Model**.

Vorticity-Based

selects the vorticity-based calculation of the deformation tensor S (see Equation 4–22 in the Theory Guide).

Strain/Vorticity-Based

selects the strain/vorticity-based calculation of the deformation tensor S (see Equation 4–24 in the Theory Guide).

Spalart-Allmaras Options

contains options for the Spalart-Allmaras model. This portion of the dialog box will appear only if **Spalart-Allmaras** is selected as the **Model**.

Low-Re Damping

allows you to enable a low-Reynolds damping to the turbulent viscosity (see *Low-Re Damping* (p. 675) for details).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

k-epsilon Model

contains options for specifying which of the k- ε models is to be used. This portion of the dialog box will appear only if **k-epsilon** is selected as the **Model**.

Standard

selects the standard k- ε model, described in Standard k- ε Model in the Theory Guide and *Setting Up the k- ε Model* (p. 658).

RNG

selects the RNG k- ε model, described in RNG k- ε Model. in the Theory Guide and *Setting Up the k- ε Model* (p. 658).

Realizable

selects the realizable k- ε model, described in Realizable k- ε Model in the Theory Guide and *Setting Up the k- ε Model* (p. 658).

RNG Options

specifies parameters that affect the solution of problems solved with the RNG k- ε model. This portion of the dialog box will appear only if **RNG** is selected as the **k-epsilon Model**.

Differential Viscosity Model

specifies whether or not the low-Reynolds-number RNG modifications to turbulent viscosity should be included. By default, this option is turned off. It is likely to have an effect only when the near-wall regions in the domain are well resolved in terms of mesh density. See *Differential Viscosity Modification* (p. 674) for details.

Swirl Dominated Flow

specifies whether or not the RNG modification to turbulent viscosity for swirling flows should be included. This option is available only in 3D and 2D axisymmetric swirl solvers, and it can yield improved predictions when solving flows with significant swirl. See *Swirl Modification* (p. 674) for details.

k-omega Model

contains options for specifying which of the k- ω models is to be used. This portion of the dialog box will appear only if **k-omega** is selected as the **Model**.

Standard

selects the standard k- ω model, described in Standard k- ω Model in the Theory Guide and *Setting Up the k- ω Model* (p. 661).

SST

selects the shear-stress transport (SST) k- ε model, described in Shear-Stress Transport (SST) k- ω Model in the Theory Guide and *Setting Up the k- ω Model* (p. 661).

k-omega Options

specifies parameters that affect the solution of problems solved with the k- ω models. This portion of the dialog box will appear only if **k-omega** is selected as the **Model**.

Low-Re Corrections

specifies whether corrections that improve the accuracy in predicting low Reynolds number flows should be included. This option is available only for the standard k- ω model. See *Low-Re Corrections* (p. 675) for details.

Shear Flow Corrections

specifies whether corrections that improve the accuracy in predicting free shear flows should be included. This option is available only for the standard k- ω model. See *Shear Flow Corrections* (p. 675) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence Damping

is required for the accurate modeling of the interfacial area. This option is available for the standard and SST k- ω model. The VOF and Mixture models and also the Eulerian multiphase model with the immiscible fluid model must be selected when using this option. Enter the desired **Damping Factor**, which by default is set to 10. See *Turbulence Damping* (p. 675) for details.

Reynolds-Stress Model

specifies the various Reynolds stress models (RSM).

Linear Pressure-Strain

enables the linear pressure-strain model. See Linear Pressure-Strain Model in the Theory Guide for details.

Quadratic Pressure-Strain

enables the quadratic pressure-strain model for superior performance in a range of basic shear flows, including plane strain, rotating plane shear, and axisymmetric expansion/contraction. See Quadratic Pressure-Strain Model in the Theory Guide for details. Note that this option cannot be used with the **Wall Reflection Effects** option or the **Enhanced Wall Treatment**.

Stress-Omega

enables a stress-transport model that is based on the omega equations and LRR model [*102*] (p. 2214). This model is ideal for modeling flows over curved surfaces and swirling flows. See Low-Re Stress-Omega Model in the Theory Guide for details.

Reynolds-Stress Options

specifies parameters that affect the solution of problems solved with the Reynolds stress model. This portion of the dialog box will appear only if **Reynolds Stress** is selected as the **Model**.

Wall BC from k Equation

enables the explicit setting of boundary conditions for the Reynolds stresses near the walls, using the values computed with Equation 4–218 in the Theory Guide. See *Solving the k Equation to Obtain Wall Boundary Conditions* (p. 676) for details. This option is on by default.

Wall Reflection Effects

enables the calculation of the component of the pressure strain term responsible for the redistribution of normal stresses near the wall. See *Including the Wall Reflection Term* (p. 676) for details. Note that this option is not available if you have enabled the **Quadratic Pressure-Strain Model**.

RANS Model

contains options for the subgrid-scale model used by the **Detached Eddy Simulation Model**. This portion of the dialog box will appear only if **Detached Eddy Simulation Model** is selected as the **Model**.

Spalart-Allmaras

enables the Spalart-Allmaras RANS model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

Realizable k-epsilon

enables the Realizable k- ε RANS model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

SST k-omega

enables the SST k- ω RANS Model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

DES Options

contain the option to include a delayed Detached Eddy Simulation.

Delayed DES

is useful for RANS meshes with high aspect ratios in the boundary layer. This option preserves the RANS model throughout the boundary layer. (See *Delayed Detached Eddy Simulation (DDES)* (p. 674) for details.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subgrid-Scale Model

contains options for the subgrid-scale model used by the LES model. This portion of the dialog box will appear only if **Large Eddy Simulation** is selected as the **Model**.

Smagorinsky-Lilly

selects the Smagorinsky-Lilly subgrid-scale model described in Subgrid-Scale Models in the Theory Guide.

WALE

selects the Wall-Adapting local Eddy-Viscosity model described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

Kinetic-Energy Transport

selects the dynamic kinetic energy subgrid-scale model described in Dynamic Kinetic Energy Subgrid-Scale Model in the Theory Guide.

LES Model Options

contains options for the Large Eddy Simulation model. This portion of the dialog box will appear only if **Large Eddy Simulation** is selected as the **Model**.

Dynamic Stress

enables the dynamic stress model. It is available when the LES option **Smagorinsky-Lilly** is enabled.

Dynamic Energy Flux

enables the dynamic energy flux model. It is available when the LES option **Kinetic-Energy Transport** is enabled.

Dynamic Scalar Flux

enables the dynamic computation of turbulent Sc (σ_t in Equation 8–5 in the Theory Guide). See Definition of the Mixture Fraction in the Theory Guide for details.

Near-Wall Treatment

specifies the near-wall treatment to be used for modeling turbulence. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for details about the available methods. This portion of the dialog box will appear if **k-epsilon** or **Reynolds Stress** is selected as the **Model**.

Standard Wall Functions

enables the use of standard wall functions (described in Standard Wall Functions in the Theory Guide).

Non-Equilibrium Wall Functions

enables the use of non-equilibrium wall functions (described in Non-Equilibrium Wall Functions in the Theory Guide).

Enhanced Wall Treatment

enables the use of the enhanced wall treatment (described in Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide). Note that this option will not appear if you have enabled the **Quadratic Pressure-Strain Model** under **Reynolds-Stress Options**.

User-Defined Wall Functions

enables you to hook a user-defined function, used to define the **Law of the Wall**. See User-Defined Wall Functions in the Theory Guide for more information.

Enhanced Wall Treatment Options

allows you to include pressure gradient or thermal effects in the calculation. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide.

Pressure Gradient Effects

enables the effect of pressure gradient.

Thermal Effects

enables thermal effects in the calculation. This option appears only if the energy equation is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

contains general options for viscous modeling.

Viscous Heating

(if enabled) includes the viscous dissipation terms in the energy equation. This option is recommended when you are solving a compressible flow. Note that this option is always turned on when one of the density-based solvers is used; you will not be able to turn it off.

Low-Pressure Boundary Slip

includes slip boundary conditions for velocity and temperature for modeling fluid flow at very low pressures as in semiconductor fabrication devices. See Slip Boundary Formulation for Low-Pressure Gas Systems in the Theory Guide. This option is available only for laminar flows.

Full Buoyancy Effects

enables the inclusion of buoyancy effects on ε. See *Including Turbulence Generation Due to Buoyancy* (p. 674) for details. This option will appear if **k-epsilon** or **Reynolds Stress** is selected as the **Model** *and* a non-zero gravitational acceleration has been specified in the *Operating Conditions Dialog Box* (p. 1806).

Turbulent Drift Force

includes the effect of drift velocity. See *Modeling Turbulence* (p. 1163). Note that this option is only available if either the **Eulerian** multiphase model is selected, or the **Mixture** model with **Slip Velocity** is enabled in the **Multiphase Model** dialog box.

Turbulence Multiphase Model

contains options for multiphase turbulence models. This portion of the dialog box will appear if **Eulerian** is selected as the **Model** in the *Multiphase Model Dialog Box* (p. 1642).

Mixture

specifies the (default) mixture turbulence model.

Dispersed

specifies the dispersed turbulence model.

Per Phase

specifies the calculation of a set of turbulence equations for each phase.

See Turbulence Models in the Theory Guide for details about the available multiphase turbulence models.

Model Constants

contains constants used in the equations for turbulence. See Spalart-Allmaras Model, Standard k-ε Model, RNG k-ε Model, k-kl-ω Transition Model, Transition SST Model, Realizable k-ε Model, Reynolds Stress Model (RSM), Standard k-ω Model, Shear-Stress Transport (SST) k-ω Model, and Large Eddy Simulation (LES) Model in the Theory Guide for details about these constants.

Cb1

(only for the Spalart-Allmaras model) is the constant C_{b1} in Equation 4–19 in the Theory Guide.

Cb2

(only for the Spalart-Allmaras model) is the constant C_{b2} in Equation 4–15 in the Theory Guide.

Cv1

(only for the Spalart-Allmaras model) is the constant C_{v1} in Equation 4–17 in the Theory Guide.

Cw2

(only for the Spalart-Allmaras model) is the constant C_{w2} in Equation 4–28 in the Theory Guide.

Cw3

(only for the Spalart-Allmaras model) is the constant C_{w3} in Equation 4–27 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cprod

(only for the Spalart-Allmaras model when the **Strain/Vorticity-Based Production** option is used) is the constant C_{prod} in Equation 4–24 in the Theory Guide.

Cmu

(only for the standard or RNG k- ε model, the RSM, or the k- kl- ω Transition model) is the constant C_μ that is used to compute μ_t.

C1-Epsilon

(only for the standard or RNG k- ε model or the RSM) is the constant $C_{1\varepsilon}$ used in the transport equation for ε.

C2-Epsilon

(only for the standard, RNG, or realizable k- ε model or the RSM) is the constant $C_{2\varepsilon}$ used in the transport equation for ε.

C3-Epsilon

(only for the dispersed or per-phase k- ε multiphase models) is the constant $C_{3\varepsilon}$ in Equation 17–248 in the Theory Guide.

C-lambda

(only for the k- kl- ω Transition model) is the constant C_λ in the definition of the effective length,

CR

(only for the k- kl- ω Transition model) is the constant C_R used in the definition of R, where R represents the averaged effect of the breakdown of streamwise fluctuations into turbulence during bypass transition

ANAT

(only for the k- kl- ω Transition model) is the constant A_{NAT}

ATS

(only for the k- kl- ω Transition model) is the constant A_{TS}

CNAT, crit

(only for the k- kl- ω Transition model) is the constant $C_{NAT,crit}$

CTS, crit

(only for the k- kl- ω Transition model) is the constant $C_{TS,crit}$

CRNAT

(only for the k- kl- ω Transition model) is the constant $C_{R,NAT}$

Anu

(only for the k- kl- ω Transition model) is the constant A_v

CINT

(only for the k- kl- ω Transition model) is the constant C_{INT}

Cw1

(only for the k- kl- ω Transition model) is the constant $C_{\omega 1}$

Cw3

(only for the k- kl- ω Transition model) is the constant $C_{\omega 3}$

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calpha-teta

(only for the k- kl- ω Transition model) is the constant $C_{\alpha,\theta}$

Ctaul

(only for the k- kl- ω Transition model) is the constant $C_{\tau,1}$

SDR Prandtl Number

(only for the k- kl- ω Transition model) is the effective "Prandtl" number for the transport of the specific dissipation rate, σ_{ω}.

Ca1

(only for the Transition SST model)

Ca2

(only for the Transition SST model)

Ce1

(only for the Transition SST model)

Ce2

(only for the Transition SST model)

C_thetat

(only for the Transition SST model)

C_s1

(only for the Transition SST model)

Intermit. Prandtl #)

(only for the Transition SST model)

Re_theta. Prandtl #)

(only for the Transition SST model)

Swirl Factor

sets the value of α_s in Equation 4–40 in the Theory Guide. This item appears for the RNG k- ε model when the **Swirl Dominated Flow** option is turned on.

Alpha*_inf

(only for the standard or SST k- ω model, and the Transition SST model) is the constant α_{∞}^{*} in Equation 4–68 in the Theory Guide.

Alpha_inf

(only for the standard or SST k- ω model, and the Transition SST model) is the constant α_{∞} in Equation 4–76 in the Theory Guide.

Alpha_0

(only for the standard or SST k- ω model with the **Transitional Flows** option enabled) is the constant α_0 in Equation 4–76 in the Theory Guide.

Beta*_inf

(only for the standard or SST k- ω model, and the Transition SST model) is the constant β_{∞}^{*} in Equation 4–81 in the Theory Guide.

Beta_i

(only for the standard k- ω model) is the constant β_i in Equation 4–89 in the Theory Guide.

R_beta

(only for the standard or SST k- ω model) is the constant R_{β} in Equation 4–81 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R_k

(only for the standard or SST k- ω model with the **Transitional Flows** option enabled) is the constant R_k in Equation 4–68 in the Theory Guide.

R_w

(only for the standard or SST k- ω model with the **Transitional Flows** option enabled) is the constant R_ω in Equation 4–76 in the Theory Guide.

Zeta*

(only for the standard or SST k- ω model) is the constant ζ^* in Equation 4–80 in the Theory Guide.

Mt0

(only for the standard or SST k- ω model) is the constant M_{t0} in Equation 4–90 in the Theory Guide.

a1

(only for the SST k- ω model, and the Transition SST model) is the constant a_1 in Equation 4–98 in the Theory Guide.

Beta_i (Inner)

(only for the SST k- ω model, and the Transition SST model) is the constant $\beta_{i,1}$ in Model Constants in the Theory Guide.

Beta_i (Outer)

(only for the SST k- ω model, and the Transition SST model) is the constant $\beta_{i,2}$ in Model Constants in the Theory Guide.

Cs

(only for LES) is the Smagorinsky constant C_s in Equation 4–252 in the Theory Guide.

C1-PS

(only for RSM) is the constant C_1 in Equation 4–189 in the Theory Guide.

C2-PS

(only for RSM) is the constant C_2 in Equation 4–190 in the Theory Guide.

C1′-PS

(only for RSM) is the constant C_1' in Equation 4–191 in the Theory Guide.

C2′-PS

(only for RSM) is the constant C_2' in Equation 4–191 in the Theory Guide.

C1-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_1 in Equation 4–200 in the Theory Guide.

C1′-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_1^* in Equation 4–200 in the Theory Guide.

C2-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_2 in Equation 4–200 in the Theory Guide.

C3-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_3 in Equation 4–200 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C3'-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_3^* in Equation 4–200 in the Theory Guide.

C4-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_4 in Equation 4–200 in the Theory Guide.

C5-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_5 in Equation 4–200 in the Theory Guide.

Prandtl Number

(only for the Spalart-Allmaras model) is the constant $\sigma_{\tilde{\nu}}$ in Equation 4–15 in the Theory Guide.

TKE Prandtl Number

(only for the standard or realizable k- ε model, the standard or SST k- ω model, the k- kl- ω Transition model, or the RSM) is the effective "Prandtl" number for transport of turbulence kinetic energy σ_k. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence kinetic energy via turbulent transport.

TKE (Inner) Prandtl #

(only for the SST k- ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of turbulence kinetic energy, $\sigma_{k,1}$, inside the near-wall region. See Modeling the Effective Diffusivity in the Theory Guide for details.

TKE (Outer) Prandtl #

(only for the SST k- ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of turbulence kinetic energy, $\sigma_{k,2}$, outside the near-wall region. See Modeling the Effective Diffusivity in the Theory Guide for details.

TDR Prandtl Number

is the effective "Prandtl" number for transport of the turbulent dissipation rate, σ_ε, for the standard or realizable k- ε model or the RSM. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence dissipation via turbulent transport.

For the standard k- ω model, the **TDR Prandtl Number** is the effective "Prandtl" number for the transport of the specific dissipation rate, σ_ω.

SDR (Inner) Prandtl #

(only for the SST k- ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of the specific dissipation rate, $\sigma_{\omega,1}$, inside the near-wall region. See Modeling the Effective Diffusivity in the Theory Guide for details.

SDR (Outer) Prandtl #

(only for the SST k- ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of the specific dissipation rate, $\sigma_{\omega,2}$, outside the near-wall region. See Modeling the Effective Diffusivity in the Theory Guide for details.

Dispersion Prandtl Number

(only for the k- ε multiphase models) is the effective "Prandtl" number for the dispersed phase, σ_{pq}. See Turbulence Models in the Theory Guide for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Energy Prandtl Number

(for any turbulence model except the RNG k- ε model) is the turbulent Prandtl number for energy, Pr_t, in Equation 4–208 in the Theory Guide. (This item will not appear for premixed or partially premixed combustion models.)

Wall Prandtl Number

(for all turbulence models) is the turbulent Prandtl number at the wall, Pr_t in Equation 4–278 in the Theory Guide. (This item will not appear for adiabatic premixed combustion or partially premixed combustion models.)

Turbulent Schmidt Number

(for turbulent species transport calculations using any turbulence model except the RNG k- ε model) is the turbulent Schmidt number, Sc_t, in Equation 7–3 in the Theory Guide.

PDF Schmidt Number

(for non-premixed or partially premixed combustion calculations using any turbulence model) is the model constant σ_t in Equation 8–5 in the Theory Guide.

User-Defined Transition Correlations

(only for the Transition SST model) allows you to select the user-defined correlations for **F_length**, **Re_thetac**, **Re_thetat**.

User-Defined Functions

allows you to select the user-defined functions for various constants.

Turbulent Viscosity

appears for Spalart Allmaras, k- ε and k- ω models. You can select the user-defined functions for turbulent viscosity in the drop-down list.

Prandtl Numbers

contains a list of relevant Prandtl numbers for which you can select user-defined functions.

TKE Prandtl Number

allows you to select a user-defined function to define the TKE Prandtl number for the standard and realizable k- ε models and the standard k- ω model.

TDR Prandtl Number

allows to select a user-defined function to define the TDR Prandtl number for the standard and realizable k- ε models.

Energy Prandtl Number

allows you to select a user-defined function to define the Energy Prandtl number for the standard and realizable k- ε models and the standard k- ω model when energy is enabled.

Wall Prandtl Number

allows you to select a user-defined function to define the Wall Prandtl number for the standard and realizable k- ε models and the standard k- ω model when energy is enabled.

SDR Prandtl Number

allows you to select a user-defined function to define the SDR Prandtl number for the standard k- ω model.

Subgrid-Scale Turbulent Viscosity

allows you to select a user-defined function for the subgrid-scale turbulent viscosity for the LES model.

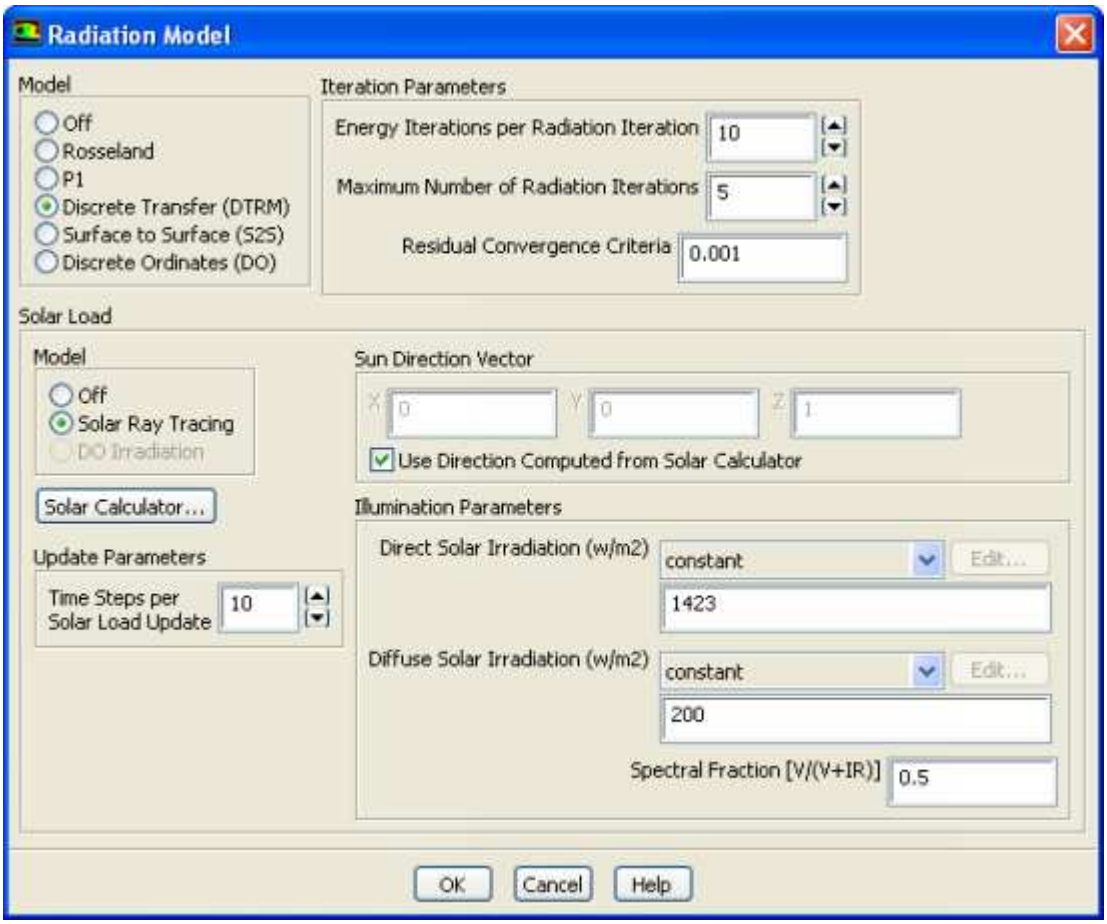
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shielding Functions

allows you to select the shielding functions (**SST F1 Function**, **SST F2 Function**, **DDES**, and **IDDES**) for the SST Detached Eddy Simulation Model (see *Shielding Functions for the SST Detached Eddy Simulation Model* (p. 678) for details).

35.3.4. Radiation Model Dialog Box

The **Radiation Model** dialog box allows you to select a model for radiation heat transfer and set the associated parameters. See *Using the Radiation Models* (p. 705) – *Defining Non-Gray Radiation for the DO Model* (p. 721) for details about the items below.



Controls

Model

indicates which model, if any, is used to calculate radiation heat transfer. See *Modeling Radiation* (p. 704) for details about modeling radiation heat transfer.

Off

disables the calculation of radiation heat transfer.

Rosseland

enables the Rosseland radiation model.

P1

enables the P-1 radiation model.

Discrete Transfer (DTRM)

enables the discrete transfer radiation model (DTRM).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface to Surface (S2S)
enables the surface-to-surface (S2S) radiation model.

Discrete Ordinates (DO)
enables the discrete ordinates (DO) radiation model.

DO/Energy Coupling
allows you to couple the energy and DO intensity equations at each cell, solving them simultaneously. See *Defining Non-Gray Radiation for the DO Model* (p. 721) for details.

Iteration Parameters
contains parameters related to the DTRM, S2S and the DO models. This portion of the dialog box will appear only if **Discrete Transfer (DTRM)**, **Surface to Surface (S2S)**, or **Discrete Ordinates (DO)** is selected as the **Model**.

Flow Iterations per Radiation Iteration
controls the frequency with which the radiation terms are updated as the solution proceeds. It is set to 10 by default. This implies that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence. This appears for the **Discrete Transfer (DTRM)** and **Discrete Ordinates (DO)** models.

Number of DTRM Sweeps
controls the number of DTRM passes.

Tolerance
allows you to control the accuracy during each iteration.

Energy Iterations per Radiation Iteration
controls the frequency with which the radiation terms are updated as the continuous phase solution proceeds. The **Energy Iterations per Radiation Iteration** parameter is set to 10 by default. This implies that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.

Maximum Number of Radiation Iterations
controls the maximum number of iterations of the radiation calculation during each global iteration. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

This item appears only when **Discrete Transfer** or **Surface to Surface** is selected as the **Model**.

Residual Convergence Criteria
determines when the radiation intensity update is converged. It is defined as the maximum normalized change in the surface intensity from one radiation iteration to the next (see *Equation 14–8* (p. 736)).

This item appears only when **Discrete Transfer** or **Surface to Surface** is selected as the **Model**.

View Factors and Clustering
contains parameters related to the S2S model. This portion of the dialog box will appear only if **Surface to Surface** is selected as the **Model**. See *Setting Up the S2S Model* (p. 709) for more information about the use of these parameters.

Settings...
opens the *View Factors and Clustering Dialog Box* (p. 1660), in which you can set parameters related to surface clusters and view factors.

Compute/Write/Read...
allows you to compute the view factors, write the computed view factors to a file, and read the file into ANSYS FLUENT. When you click **Compute/Write/Read...**, the *The **Select File** Dialog Box* (p. 33)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will open so that you can specify a name for the file where ANSYS FLUENT should save the view factors after computing them.

Read Existing File...
opens the *The **Select File** Dialog Box* (p. 33), in which you can specify the file from which ANSYS FLUENT should read view factors.

Angular Discretization
contains parameters for angular discretization and pixelation for the DO model. This portion of the dialog box will appear only if **Discrete Ordinates** is selected as the **Model**. See *Setting Up the DO Model* (p. 720) for more information about the use of these parameters.

Theta Divisions, Phi Divisions
define the number of control angles used to discretize each octant of the angular space (see Figure 5.3: "Angular Coordinate System" in the Theory Guide).

Theta Pixels, Phi Pixels
are used to control the pixelation that accounts for any control volume overhang (see Figure 5.7: "Pixelation of Control Angle" in the Theory Guide and the figures and discussion preceding it).

Non-Gray Model
contains parameters related to the non-gray P-1 model or non-gray DO model. This portion of the dialog box will appear only if **P1** or **Discrete Ordinates** is selected as the **Model**. See *Setting Up the P-1 Model with Non-Gray Radiation* (p. 706) or *Defining Non-Gray Radiation for the DO Model* (p. 721) for more information about the use of these parameters.

Number of Bands
specifies the number of bands for the non-gray radiation.

Wavelength Intervals (n=1)
contains inputs that define each wavelength band. (It appears only when the **Number of Bands** is non-zero.) For each band, you can specify a **Name**, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum ($n = 1$). ANSYS FLUENT will automatically account for the refractive index in setting band limits for media with n different from unity.

Solar Load
contains parameters related to solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain. It is available only for the 3D version of ANSYS FLUENT. See *Solar Load Model* (p. 740) for more information about the use of these parameters.

Model
indicates which model is used to calculate radiation effects.

Off
disables the calculation of solar radiations.

Solar Ray Tracing
enables the solar ray tracing algorithm

DO Irradiation
enables the DO irradiation option. Select **Discrete Ordinates** under **Model** before selecting this option.

Solar Calculator
opens the *Solar Calculator Dialog Box* (p. 1665).

Sun Direction Vector
contains the components of sun direction vector.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X, Y, Z
are the components of the sun direction vector.

Use Direction Computed from Solar Calculator
enables the use of direction vector computed from the solar calculator.

Illumination Parameters
contains illumination options.

Direct Solar Irradiation
is the amount of energy per unit area due to direct solar irradiation.

Diffuse Solar Irradiation
is the amount of energy per unit area due to diffuse solar irradiation.

Spectral Fraction
is the fraction of incident solar radiation in the visible part of the solar radiation spectrum. The spectral fraction is not used for DO irradiation since the DO implementation is intended only for a single band. This parameter is available only for **Solar Ray Tracing**.

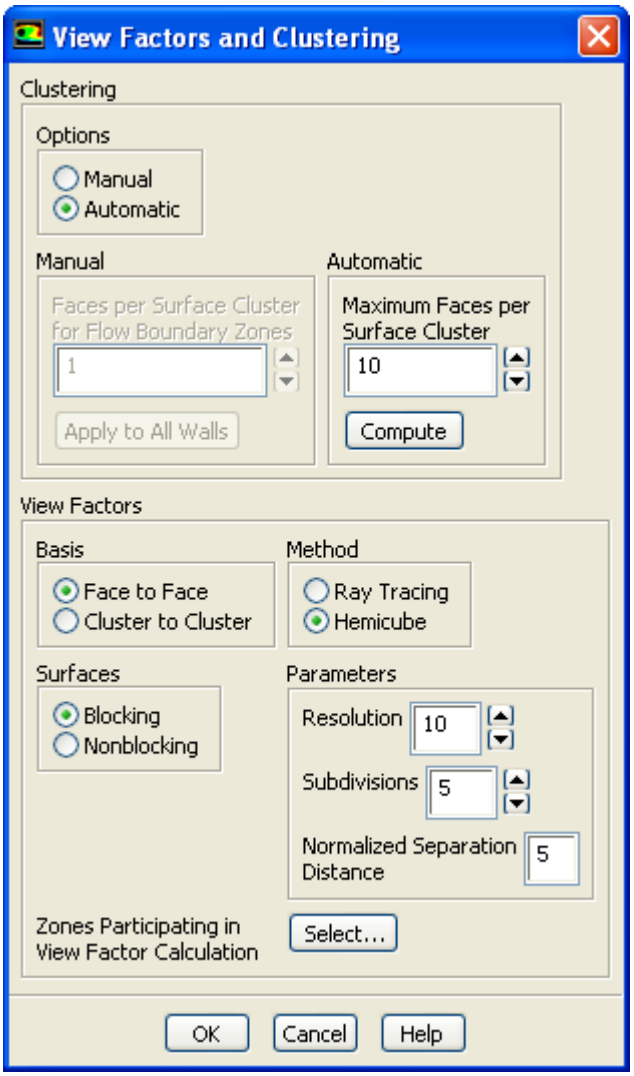
Update Parameters
contains update parameters for transient simulations.

Time Steps per Solar Load Update
specifies the number of time steps that will direct the ANSYS FLUENT solver to update the solar load data for the specified flow-time intervals in the unsteady solution process.

35.3.5. View Factors and Clustering Dialog Box

The **View Factors and Clustering** dialog box allows you to set parameters related to surface clusters and view factors for the surface-to-surface radiation model. See *Setting Up the S2S Model* (p. 709) for information about using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Clustering

contains the methods and settings for creating surface clusters. See *Forming Surface Clusters* (p. 711) for information about surface cluster settings.

Options

gives you a choice of forming clusters either manually or automatically.

Manual

allows you to form clusters manually.

Automatic

allows ANSYS FLUENT to form the cluster automatically.

Manual

(available only when the **Manual** option is enabled) allows you to set the faces per surface cluster (FPSC) value for walls and inlet and outlet boundaries.

Faces per Surface Cluster for Flow Boundary Zones

specifies the number of faces in each surface cluster for inlet and outlet boundaries (i.e., exhaust fan, inlet vent, intake fan, outlet vent, mass-flow inlet, pressure far-field, pressure inlet, pressure outlet, outflow, and velocity inlet boundaries). This FPSC value can also be applied to all walls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that are adjacent to fluid zones by clicking the **Apply to All Walls** button. The default is set to 1. See Clustering in the Theory Guide for details about clustering.

Apply to All Walls
applies the value specified in the **Faces per Surface Cluster for Flow Boundary Zones** field to all walls that are adjacent to fluid zones.

Automatic
(available only when the **Automatic** option is enabled) allows you to assign different faces per surface cluster (FPSC) values to the walls automatically, based on the distance of the walls from and the FPSC values of the walls that are defined as critical.

Maximum Faces per Surface Cluster
specifies the maximum number of faces per surface cluster automatically assigned to non-critical wall zones adjacent to fluid zones. The default is set to 10.

Compute
results in ANSYS FLUENT automatically calculating and updating the face per surface cluster values in the boundary conditions dialog box for non-critical wall zones adjacent to fluid zones, without computing the clusters.

View Factors
contains settings for computing the view factors. See *Setting Up the View Factor Calculation* (p. 713) for information about view factor calculation settings.

Basis
specifies how surfaces are defined for the calculation of view factors. See *Selecting the Basis for Computing View Factors* (p. 713) for more information.

Face to Face
specifies that the surfaces used to calculate the view factors are the boundary faces.

Cluster to Cluster
specifies that the surfaces used to calculate the view factors are the clusters defined by the settings in the **Clustering** group box. The cluster to cluster basis is only available for 3D cases.

Surfaces
specifies the geometric orientation of surface pairs with respect to each other when using the hemicube method.

Blocking
specifies that the view factor calculation accounts for surfaces that block the views between the surfaces under consideration.

Nonblocking
specifies that the view factor calculation does not account for surfaces that block the views between the surfaces under consideration.

Method
specifies the method for computing the view factors. See *Selecting the Method for Computing View Factors* (p. 714) for information about choosing a method.

Hemicube
specifies the use of the hemicube method for computing the view factors. The hemicube method is available only for 3D and axisymmetric cases.

Ray Tracing

specifies the use of the ray tracing method for computing the view factors. For 2D cases, the number of rays used with this method is two times the value set for **Resolution** in the **Parameters** group box; for 3D cases, the number of rays is three times the square of the **Resolution** value.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The ray tracing method should not be used when any of the zones are defined as periodic. The ray tracing method is only available with the face to face basis.

Parameters

contains settings related to the hemicube and ray tracing methods for computing the view factors. All of these inputs are available when **Hemicube** is selected under **Method**, whereas only **Resolution** is available when **Ray Tracing** is selected. See *Selecting the Method for Computing View Factors* (p. 714) for information about setting the method parameters.

Resolution

specifies the resolution of the hemicube. The default value is set to `10`. You can increase the value to reduce aliasing effects that can lead to overestimated or underestimated view factors.

Subdivisions

specifies the number of subfaces into which each face is divided. The default value is set to 5. This parameter is only available when the hemicube method is used in conjunction with the face to face basis.

Normalized Separation Distance

specifies the ratio of the minimum face separation to the effective diameter of the face. The default value is set to 5. This parameter is only available for the hemicube method.

Zones Participating in View Factor Calculation

allows you to define which boundary zones participate in the view factor calculation. See *Specifying Boundary Zone Participation* (p. 715) for more information.

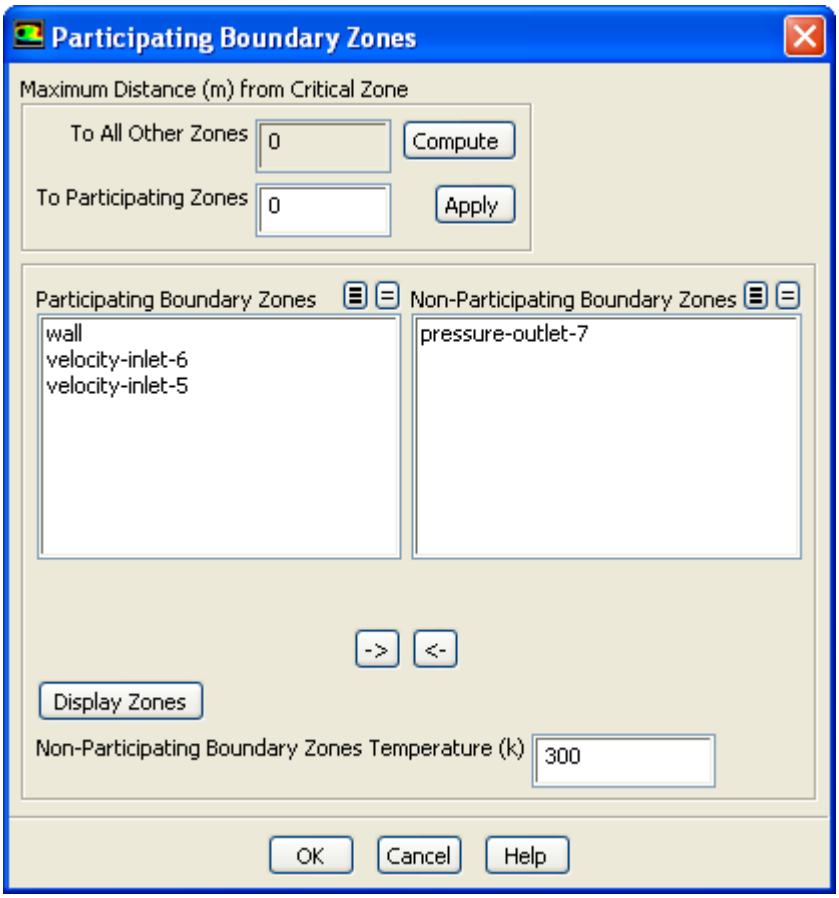
Select...

opens the *Participating Boundary Zones Dialog Box* (p. 1663), where you can define which boundary zones participate in the view factor calculation.

35.3.6. Participating Boundary Zones Dialog Box

The **Participating Boundary Zones** dialog box allows you to define which boundary zones participate in the view factor calculation, to display zones in the graphics window, and set the temperature of zones that do not participate in the view factor calculation. See *Specifying Boundary Zone Participation* (p. 715) for details. This dialog box opens when you click on the **Select...** button next to the **Zones Participating in View Factor Calculation** label in the **View Factors and Clustering** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Maximum Distance from Critical Zone

allows you to view the maximum distance between critical zones and other zones, and set all boundary zones that are located beyond a certain distance from a critical zone as not participating in the view factor calculation.

To All Other Zones

displays the maximum of the distances between the centroids of critical and all other wall, inlet, and exit zones when the **Compute** button is clicked. This field is not editable. This is only available when using the **Automatic** option for clustering.

Compute

calculates the distances between the centroids of critical zones and all the other wall, inlet, and exit zones and displays the maximum value in the **To All Other Zones** field and the **To Participating Zones** text-entry box. It requires the definition of the critical zone.

To Participating Zones

displays the maximum of the distances between the centroids of critical and all other wall, inlet, and exit zones when the **Compute** button is clicked. Unlike the **To All Other Zones** field, you can edit this text-entry box. You can specify the maximum distance allowed between the centroids of critical zones and all other wall, inlet, or exit zones that participate in the view factor calculation; when you click the **Apply** button, all of the zones will be marked as either participating or not participating, according to the distance criteria you specified. Note that this field is only available when using the **Automatic** option for clustering.

Apply

marks all of the wall, inlet, and exit zones as either participating or not participating in the view factor calculation, depending on whether the distance from their centroid to the centroid of a critical

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

zone is equal to or less than value entered for **To Participating Zones**. The **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists will be updated accordingly.

Participating Boundary Zones
shows all the zones that are participating in the view factor calculation. You can select a zone and click the arrow button that points to the left to move the zone to the **Non-Participating Boundary Zones** list.

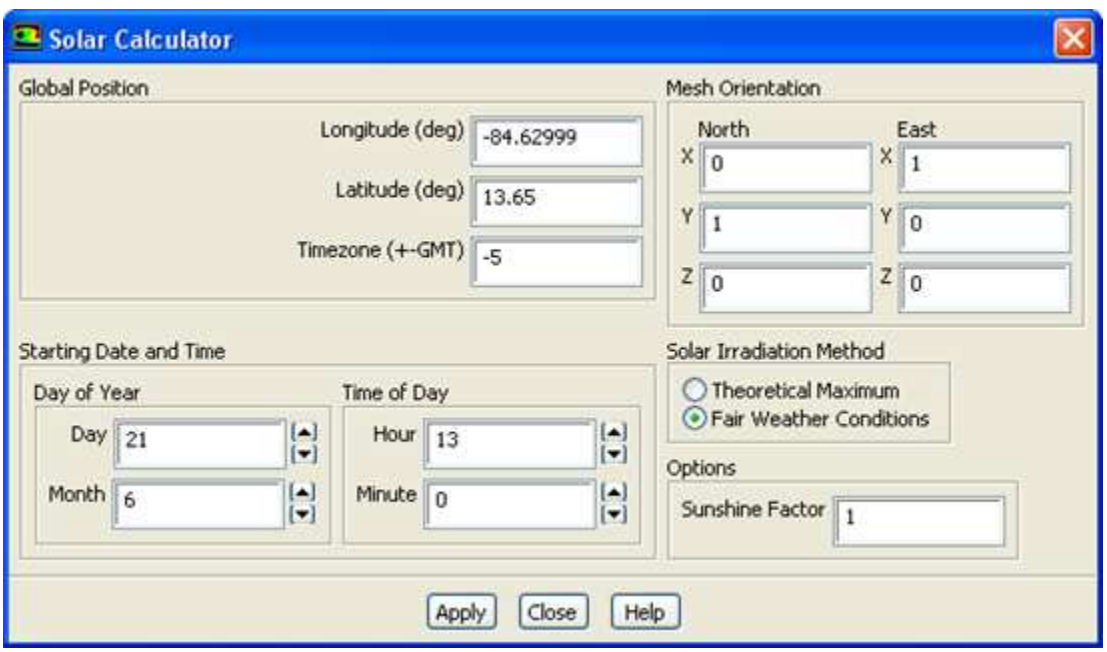
Non-Participating Boundary Zones
shows all the zones that are not participating in the view factor calculation. You can select a zone and click the arrow button that points to the right to move the zone to the **Participating Boundary Zones** list.

Display Zones
displays any zones selected in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists in the graphics window.

Non-Participating Boundary Zones Temperature
allows you to specify the temperature of the zones that do not participate in the view factor calculation.

35.3.7. Solar Calculator Dialog Box

The **Solar Calculator** dialog box allows you to set parameters related to the calculation of solar load models. See *Solar Load Model* (p. 740) for details. This dialog box opens when you click on the **Solar Calculator...** button in the **Radiation Model** dialog box.



Controls

Global Position
contains the parameters to define the position of solar radiation.

Longitude
specifies the longitude of the desired location in degrees. Values may range from -180 to 180, where negative values indicate the Western hemisphere and positive values indicate the Eastern hemisphere.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Latitude
specifies the latitude of the desired location in degrees. Values may range from -90 (South pole) to 90 (North pole) with 0 defined as the equator.

Timezone
specifies the local time zone of the desired location in hours relative to Greenwich Mean Time (+-GMT). This integer value can range from 12 to -12.

Starting Date and Time
contains parameters to specify date and time.

Day of Year
contains parameters to specify day and month.

Time of Day
contains parameters to specify hour and minutes.

Mesh Orientation
specifies the orientation as the vectors for North and East in the CFD grid system of coordinates.

Solar Irradiation Method
contains parameters to choose the solar irradiation method.

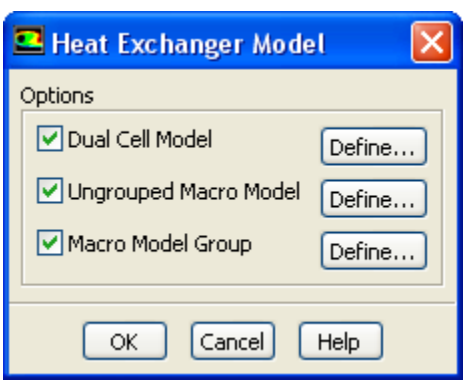
Theoretical Maximum
enables theoretically maximum solar irradiation method.

Fair Weather Conditions
enables fair weather condition solar irradiation method.

Sunshine Factor
is a linear reduction factor for the computed incident load that allows for cloud cover to be accounted for, if appropriate.

35.3.8. Heat Exchanger Model Dialog Box

The **Heat Exchanger Model** dialog box allows you to define a heat exchanger as part of your model. See *Modeling Heat Exchangers* (p. 769) for details about using the items below.



Controls

Options
contains options related to the heat exchanger model.

Dual Cell Model
allows you to enable or disable the Dual Cell heat exchanger model. When this option is enabled, the corresponding **Define...** button opens the *Dual Cell Heat Exchanger Dialog Box* (p. 1667).

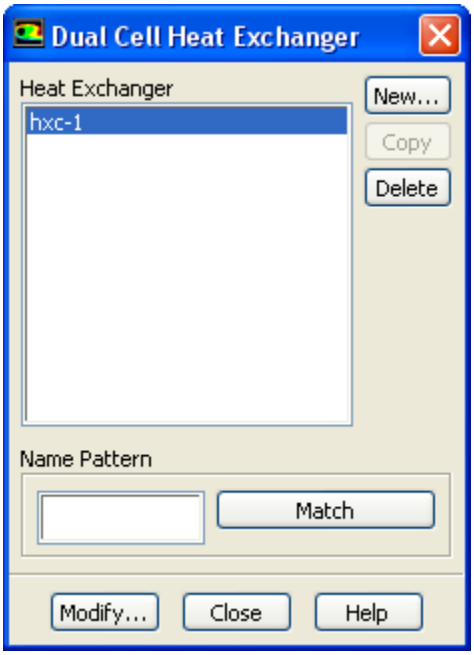
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ungrouped Macro Model
allows you to enable or disable the Ungrouped Macro heat exchanger model. When this option is enabled, the corresponding **Define...** button opens the *Ungrouped Macro Heat Exchanger Dialog Box* (p. 1672).

Macro Model Group
allows you to enable or disable the Macro heat exchanger group model. When this option is enabled, the corresponding **Define...** button opens the *Macro Heat Exchanger Group Dialog Box* (p. 1677).

35.3.9. Dual Cell Heat Exchanger Dialog Box

The **Dual Cell Heat Exchanger** dialog box allows you to use the NTU method for heat transfer calculations. This model allows the solution of auxiliary flow on a separate mesh (other than the primary fluid mesh). See *Using the Dual Cell Heat Exchanger Model* (p. 787) for information about using this dialog box.



Controls

Heat Exchanger
contains a list of predefined heat exchangers.

New...
opens the *Set Dual Cell Heat Exchanger Dialog Box* (p. 1668).

Copy
opens the *Copy From Dialog Box* (p. 1672), which allows you to copy the currently selected heat exchanger.

Delete
removes the currently selected heat exchanger.

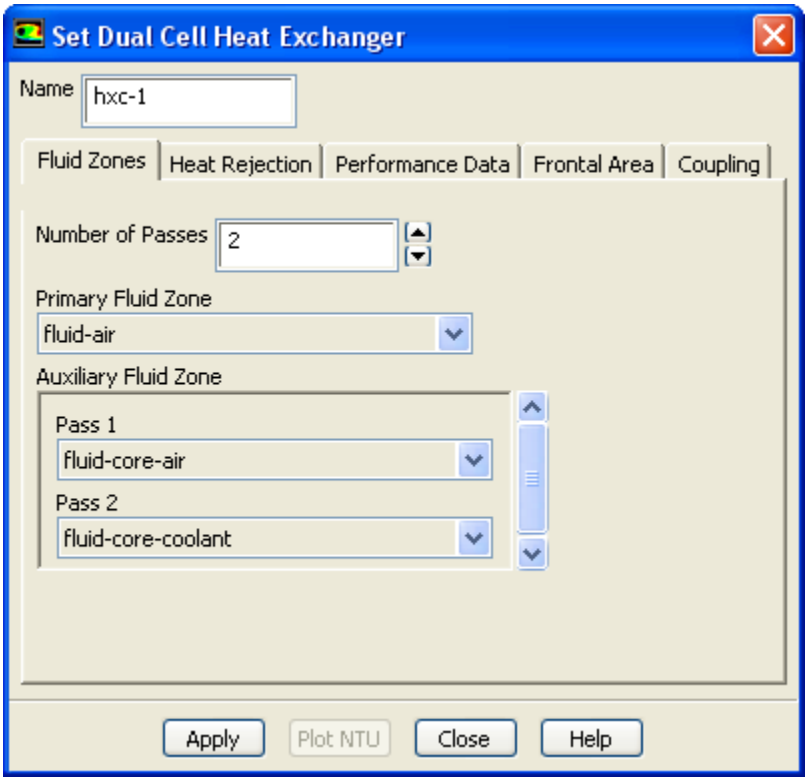
Name Pattern
specifies the pattern to look for in the names of heat exchangers. Type the pattern in the text field and click **Match** to select (or deselect) the listing in the **Heat Exchanger** list with names that match the specified pattern.

Modify...
opens the *Set Dual Cell Heat Exchanger Dialog Box* (p. 1668).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3.10. Set Dual Cell Heat Exchanger Dialog Box

The **Set Dual Cell Heat Exchanger** dialog box allows you to define the heat exchanger parameters. See *Using the Dual Cell Heat Exchanger Model* (p. 787) for details about using this dialog box.



Controls

Name
allows you to specify a name for the dual cell heat exchanger.

Fluid Zones
allows you to specify the fluid zone parameters for the heat exchanger.

Number of Passes
specifies the number of passes for the heat exchanger.

Primary Fluid Zone
allows you to specify the fluid of the primary fluid zone for the heat exchanger.

Auxiliary Fluid Zone
allows you to specify the fluid for the auxiliary fluid zone, per pass.

Important

The selected zones must be overlapping in physical space.

Heat Rejection
contains parameters specific to the rejection of heat in the heat exchanger.

Options
allows you to specify how the heat rejection in the heat exchanger is computed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fixed Heat Rejection
allows you to specify heat rejection parameters.

Fixed Inlet Temperature
allows you use total heat rejection as the desired output.

Heat Rejection Targeted
allows you to specify the heat rejection desired from the heat exchanger (available only when the **Fixed Heat Rejection** option is enabled).

Inlet Zone for Temperature Updates
allows ANSYS FLUENT to change the temperature of the specified inlet zone in order to match the targeted heat rejection (available only when the **Fixed Heat Rejection** option is enabled).

Temperature Update Under-Relaxation
controls convergence (available only when the **Fixed Heat Rejection** option is enabled).

Iteration Interval Between Temperature Updates
controls divergence (available only when the **Fixed Heat Rejection** option is enabled).

Performance Data
contains parameters for specifying the heat exchanger's performance data.

Options
allows you to choose between using raw performance data or NTU performance data.

Raw Data
allows you to specify raw performance data for the heat exchanger.

NTU Data
allows you to specify NTU performance data for the heat exchanger.

Heat Exchanger Performance Data
contains parameters concerning the heat exchanger's performance data.

NTU Table
opens the *NTU Table Dialog Box* (p. 1671) (available only when the **NTU Data** option is enabled).

Heat Transfer Table
opens the *Heat Transfer Data Table Dialog Box* (p. 1670) (available only when the **Raw Data** option is enabled).

Effectiveness-NTU Relation
computes the NTU values from the heat transfer data. Choose **cross-flow-unmixed**, **parallel-flow**, or **counter-flow**, all of which are described in NTU Relations (available only when the **Raw Data** option is enabled).

Reference Inlet Temperature
allows you to specify the inlet reference temperature for the primary and the auxiliary fluids. (available only when the **Raw Data** option is enabled).

Frontal Area
allows you to specify the frontal area for the heat exchanger.

Primary Fluid
allows you to specify a value for the **Core Frontal Area** for the primary fluid, or to compute the value from a surface zone using the **Compute From** drop-down list.

Auxiliary Fluid
allows you to specify a value for the **Core Frontal Area** for the auxiliary fluid, or to compute the value from a surface zone using the **Compute From** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupling

specifies parameters when you want to couple the heat exchanger passes.

Temperature

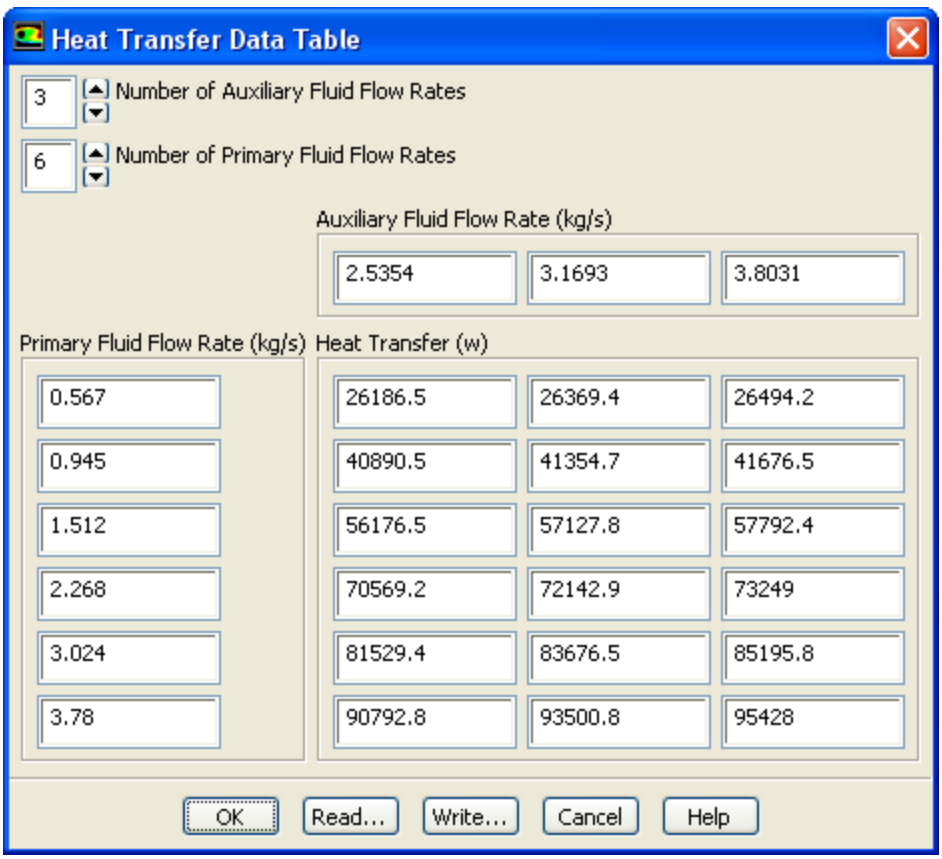
specifies, by default, the mass-weighted average for the temperature of the outlet of Pass 1 to the inlet of Pass 2. Similarly, the mass-weighted-average temperature of the outlet of Pass 2 will be applied at the inlet zone of Pass 3, and so on. (available only when multiple passes are specified in the **Fluid Zones** tab).

Plot NTU

plots the performance data curve for the selected heat exchanger. The performance data is supplied through the **Performance Data** tab.

35.3.11. Heat Transfer Data Table Dialog Box

The **Heat Transfer Data Table** dialog box contains information on the number of fluid flow rates and heat transfer data for the primary and auxiliary fluids. See *Using the Ungrouped Macro Heat Exchanger Model* (p. 771) for details about using this dialog box.



Controls

Number of Auxiliary Fluid Flow Rates

sets the number of auxiliary fluid flow rates.

Number of Primary Fluid Flow Rates

sets the number of primary fluid flow rates.

Auxiliary Fluid Flow Rate

sets fluid flow rates for the auxiliary fluid.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Primary Fluid Flow Rate
sets fluid flow rates for the primary fluid.

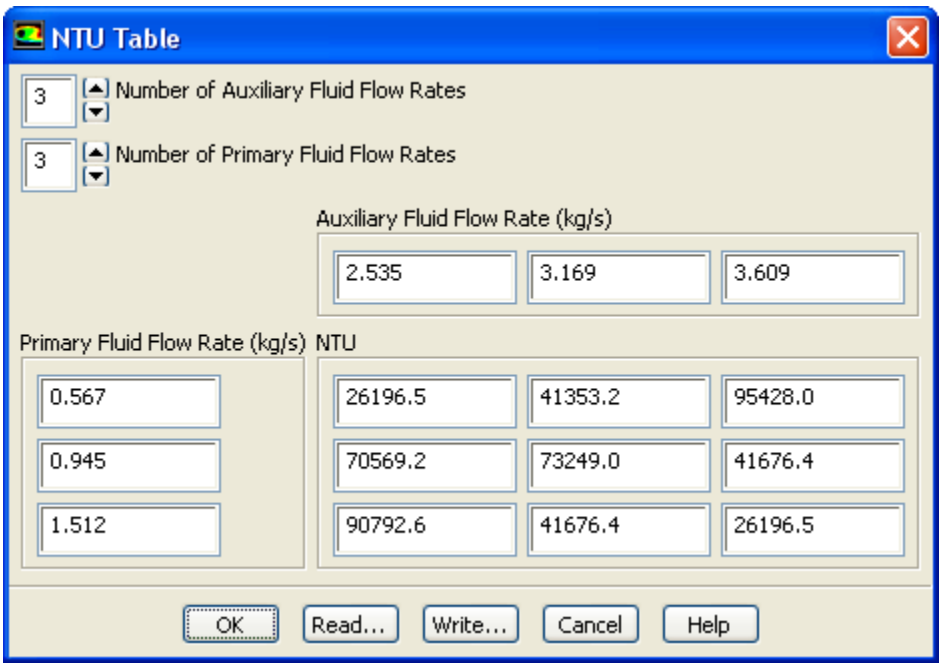
Heat Transfer
sets the heat transfer for the corresponding primary and auxiliary fluid flow rates.

Read...
allows you to read in a file containing heat transfer data.

Write...
allows you to write a file containing heat transfer data.

35.3.12. NTU Table Dialog Box

The **NTU Table** dialog box contains information on the number of fluid flow rates and NTU data for the primary and auxiliary fluids. See *Using the Ungrouped Macro Heat Exchanger Model* (p. 771) for details about using this dialog box.



Controls

Number of Auxiliary Fluid Flow Rates
sets the number of auxiliary fluid flow rates.

Number of Primary Fluid Flow Rates
sets the number of primary fluid flow rates.

Auxiliary Fluid Flow Rate
sets fluid flow rates for the auxiliary fluid.

Primary Fluid Flow Rate
sets fluid flow rates for the primary fluid.

NTU
sets the NTU values for the corresponding primary and auxiliary fluid flow rates.

Read...
allows you to read in a file containing NTU data.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write...
allows you to write a file containing NTU data.

35.3.13. Copy From Dialog Box

The **Copy From** dialog box allows you to copy the setup of one heat exchanger to another.

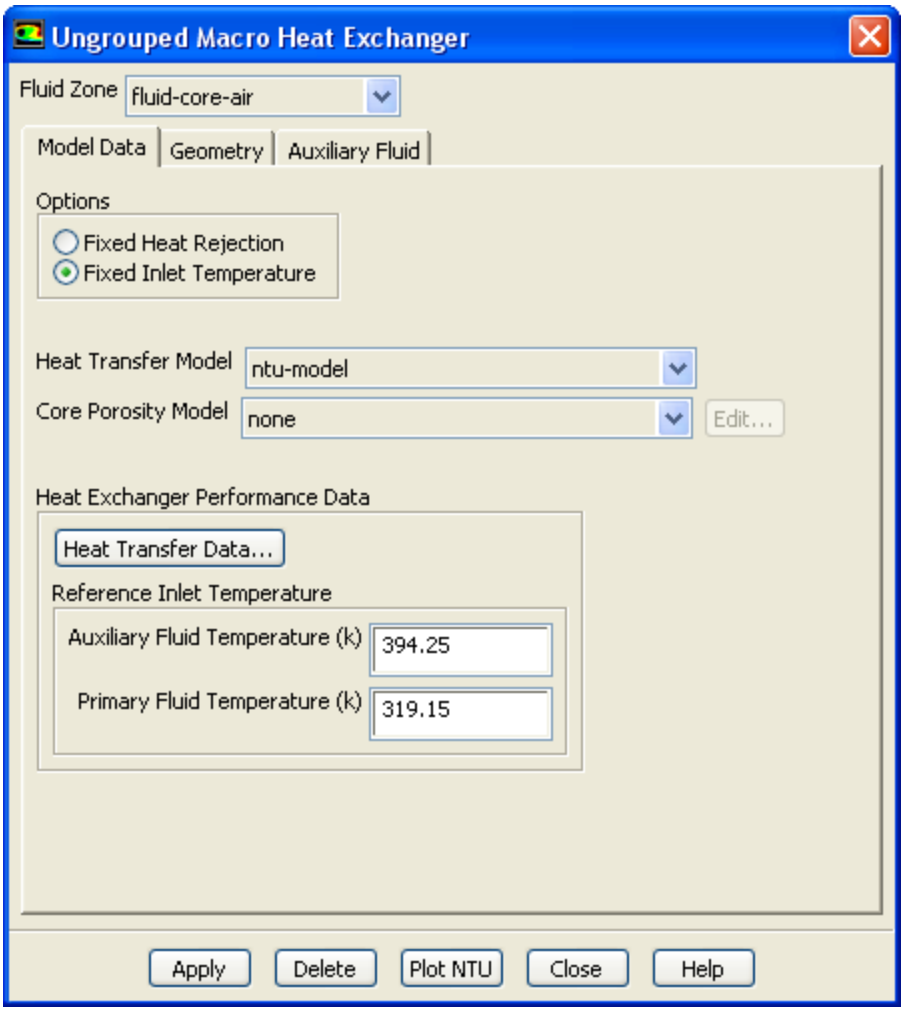
Controls

Existing Heat Exchangers
contains a list of heat exchangers, from which you can copy the settings from one heat exchanger to another.

35.3.14. Ungrouped Macro Heat Exchanger Dialog Box

The **Ungrouped Macro Heat Exchanger** dialog box allows you to set up the ungrouped macro heat exchanger model. See *Using the Ungrouped Macro Heat Exchanger Model* (p. 771) for details about using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Fluid Zone

specifies the zone that represents the heat exchanger.

Model Data

contains the parameters related to the heat exchanger model.

Options

allows you to choose one of the following settings:

Fixed Heat Rejection

specifies that ANSYS FLUENT should compute the auxiliary fluid inlet temperature for a specified heat rejection.

Fixed Inlet Temperature

specifies that ANSYS FLUENT should compute the total heat rejection of the core for a given inlet auxiliary temperature.

Heat Transfer Model

allows you to specify either the **ntu-model** or the **simple-effectiveness-model** for heat transfer.

Core Porosity Model

contains a drop-down list of all available core porosity models.

Edit...

opens the *Core Porosity Model Dialog Box* (p. 1676).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Exchanger Performance Data
contains the parameters for heat transfer.

Heat Transfer Data...
opens the *Heat Transfer Data Table Dialog Box* (p. 1670). This item will appear for the NTU model only.

Auxiliary Fluid Temperature
specifies the auxiliary fluid temperature. This item will appear only for the NTU model.

Primary Fluid Temperature
specifies the gas stream temperature. This item will appear only for the NTU model.

Velocity Effectiveness Curve...
opens the *Velocity Effectiveness Curve Dialog Box* (p. 1675) in which you can define the effectiveness of the heat exchanger core (ε in Equation 6–11 in the Theory Guide). This item will appear for simple effectiveness model only.

Geometry
contains parameters to define the macro grid.

Width
sets the width of the heat exchanger core. The **Width** is measured in the pass-to-pass direction.

Height
sets the height of the heat exchanger core. The **Height** is measured in the auxiliary fluid inlet direction.

Depth
sets the depth of the heat exchanger core.

Number of Passes
specifies the number of passes for the macro grid. (See *Figure 15.6* (p. 777).)

Number of Rows/Pass, Number of Columns/Pass
specify the number of macro rows and columns per pass in the macro grid. (See *Figure 15.6* (p. 777).)

View Passes
displays the macro grid. (This button becomes available after you click **Apply**.) The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

Draw Mesh
toggles between displaying and not displaying the mesh when the macro mesh is displayed (using the **View Passes** button). The *Mesh Display Dialog Box* (p. 1635) opens when **Draw Mesh** is selected.

Update from Plane Tool
updates the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** from the plane tool orientation. The **Width**, **Height**, and **Depth** will also be updated. See *Using the Plane Tool* (p. 1367) for information about using the plane tool.

Auxiliary Fluid Inlet Direction (height)
specifies the direction in which the auxiliary fluid enters the heat exchanger. See *Figure 15.6* (p. 777).

Pass-to-Pass Direction (width)
specifies the direction in which the auxiliary fluid moves at the end of each pass through the heat exchanger. See *Figure 15.6* (p. 777).

Auxiliary Fluid
contains the option to specify the auxiliary fluid properties.

Auxiliary Fluid Properties Method
contains options for specifying auxiliary fluid properties.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constant-specific-heat
allows you to specific a constant value for the auxiliary fluid specific heat.

user-defined-enthalpy
allows you to specify a user-defined function for the auxiliary fluid enthalpy.

Auxiliary Fluid Specific Heat
specifies the value of $c_{p,auxiliary}$ in Equation 6–17 in the Theory Guide. This value is specified only if **constant-specific-heat** is selected.

Auxiliary Fluid Enthalpy UDF
allows you to specify a UDF for the auxiliary fluid enthalpy (see Equation 6–17 in the Theory Guide). This option is available only when **user-defined-enthalpy** is selected.

Auxiliary Fluid Flow Rate
sets the flow rate of the auxiliary fluid ($\dot{m}$ in Equation 6–16 in the Theory Guide).

Heat Rejection
sets the total heat rejection (q_{total} in Equation 6–15 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Initial Temperature
sets an initial guess for the inlet temperature (T_{in} in Equation 6–11 and Equation 6–16 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Inlet Temperature
sets the auxiliary fluid initial temperature (T_{in} in Equation 6–11 and Equation 6–16 in the Theory Guide). This value is specified only if **Fixed Inlet Temperature** is selected in the **Model Data** tab.

Inlet Pressure
sets the auxiliary fluid inlet pressure. This value is specified only if **user-defined-enthalpy** is selected.

Inlet Quality
specifies the value of x in Equation 6–20 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Pressure Drop
specifies the value of p_{in} in Equation 6–21 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

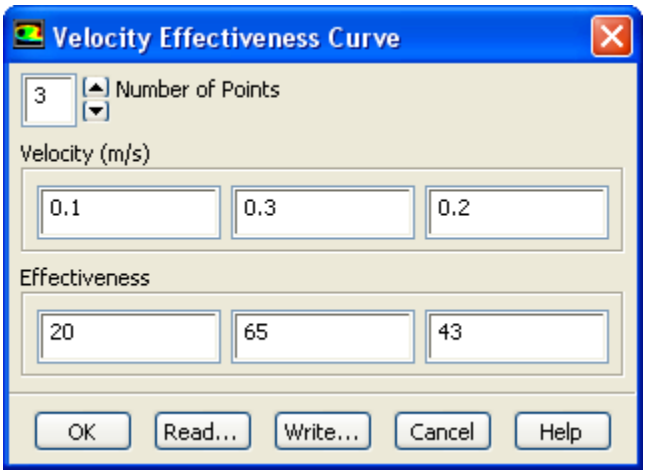
Apply
saves all the settings for the heat exchanger specified in the **Fluid Zone** list.

Delete
deletes the heat exchanger specified in the **Fluid Zone** list.

35.3.15. Velocity Effectiveness Curve Dialog Box

The **Velocity Effectiveness Curve** dialog box allows you to define effectiveness curve. It is opened by clicking **Velocity Effectiveness Curve...** in the *Ungrouped Macro Heat Exchanger Dialog Box* (p. 1672).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Points

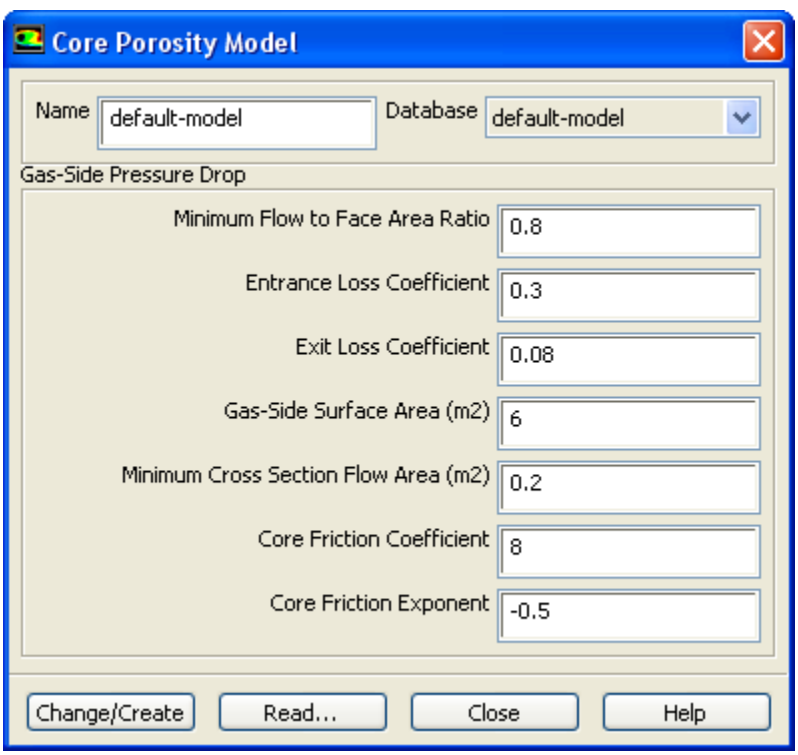
specifies the number of data pairs in the effectiveness profile. The default value of 1 indicates a constant effectiveness.

Velocity, Effectiveness

specify the data pairs for the effectiveness profile. These items are available only if the simple effective model has been selected.

35.3.16. Core Porosity Model Dialog Box

The **Core Porosity Model** dialog box allows you to modify or define a heat exchanger core model. This dialog box opens when you click **Edit...** in the *Ungrouped Macro Heat Exchanger Dialog Box* (p. 1672).



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
specifies the name of a new heat exchanger core model.

Database
contains a drop-down list of all heat exchanger core models that are currently available.

Gas-Side Pressure Drop
contains parameters that define the air-side pressure drop.

Minimum Flow to Face Area Ratio
sets the value of σ in Equation 6–2 in the Theory Guide.

Entrance Loss Coefficient
sets the value of K_c in Equation 6–2 in the Theory Guide.

Exit Loss Coefficient
sets the value of K_e in Equation 6–2 in the Theory Guide.

Gas Side Surface Area
sets the value of A in Equation 6–2 in the Theory Guide.

Minimum Cross Section Flow Area
sets the value of A_c in Equation 6–2 in the Theory Guide.

Core Friction Coefficient
sets the value of a in Equation 6–3 in the Theory Guide.

Core Friction Exponent
sets the value of b in Equation 6–3 in the Theory Guide.

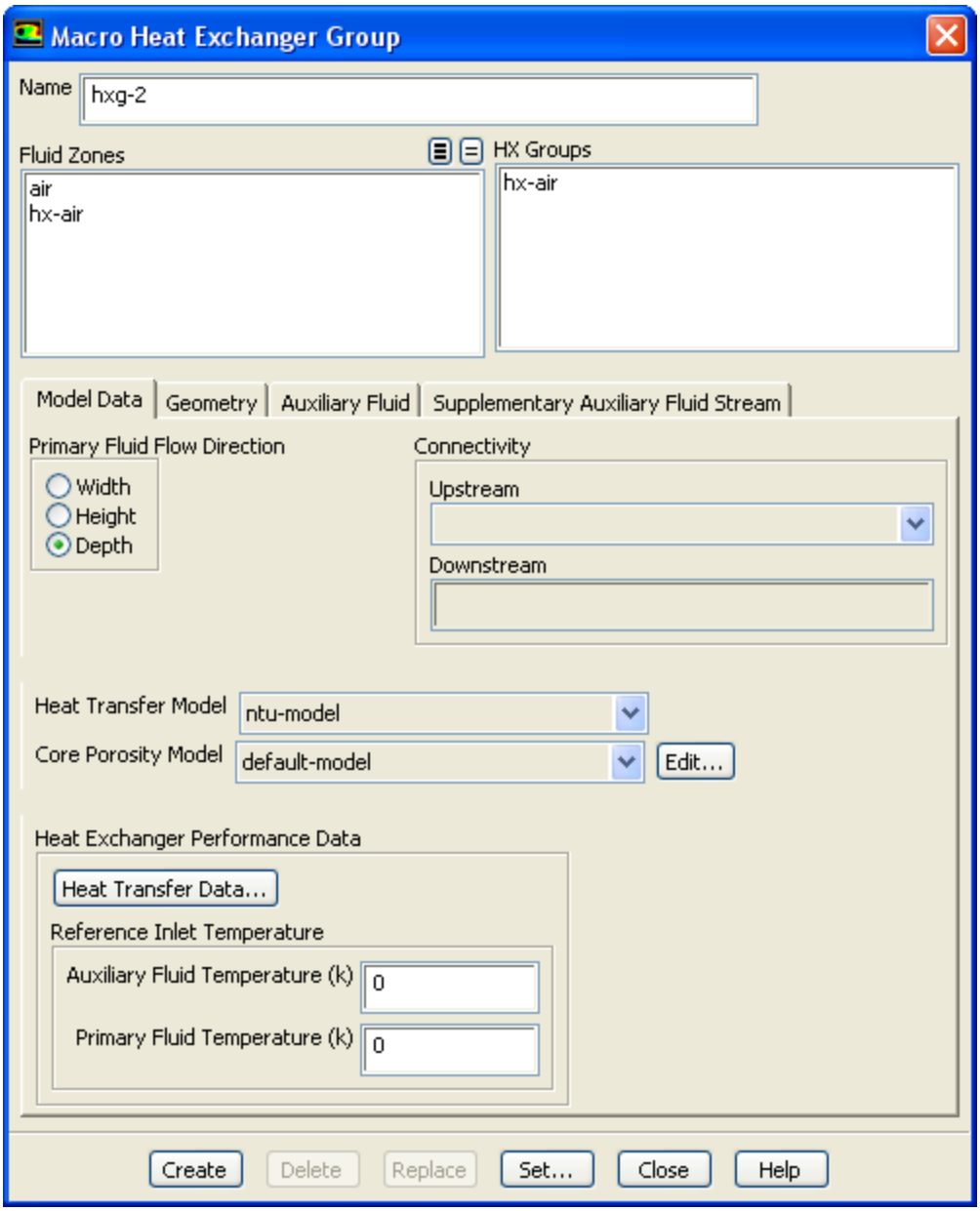
Change/Create
saves the settings in the dialog box and adds the new model to the **Database** list.

Read...
opens the *The **Select File** Dialog Box* (p. 33), in which you can select an external file containing a predefined heat exchanger core model.

35.3.17. Macro Heat Exchanger Group Dialog Box

The **Macro Heat Exchanger Group** dialog box allows you to modify or define a heat exchanger core group.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of a new heat exchanger group.

Fluid Zones

contains a list of all the fluid zones.

HX Groups

contains list of the heat exchanger groups.

Model Data

contains all the parameters to be specified for the model.

Primary Fluid Flow Direction

gives you a choice of gas flow direction.

Width, Height, Depth

specifies the width, height and the depth of the gas flow direction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Connectivity
allows you to define the upstream and downstream connections.

Upstream
specifies the upstream heat exchanger group.

Downstream
specifies the downstream heat exchanger group.

Heat Transfer Model
allows you to select either the **simple-effectiveness-model** or the **ntu-model**.

Core Porosity Model
specifies whether default values are chosen for the core porosity model.

Edit...
opens the *Core Porosity Model Dialog Box* (p. 1676) for the definition of a new core porosity model.

Heat Exchanger Performance Data
contains the parameters for heat transfer.

Heat Transfer Data...
opens the *Heat Transfer Data Table Dialog Box* (p. 1670). This dialog box allows you to define the heat transfer for different primary and auxiliary fluid flow rates. This item will appear for the NTU model only.

Velocity Effectiveness Curve...
opens the *Velocity Effectiveness Curve Dialog Box* (p. 1675)in which you can define the effectiveness of the heat exchanger core (ε in Equation 6–11 in the Theory Guide). This item will appear for simple effectiveness model only.

Auxiliary Fluid Temperature
specifies the auxiliary fluid temperature. This item will appear for the NTU model only.

Primary Fluid Temperature
specifies the gas stream fluid temperature. This item will appear for the NTU model only.

Geometry
contains parameters to define the macro grid.

Width, Height, Depth
specifies the width, height and the depth of the heat exchanger.

Number of Passes
specifies the number of passes.

Number of Rows/Pass
specifies the number of rows per pass.

Number of Columns/Pass
specifies the number of columns per pass.

View Passes
displays the macro grid. (This button becomes available after you click **Set**.) The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

Draw Mesh
toggles between displaying and not displaying the mesh when the macro mesh is displayed (using the **View Passes** button). The *Mesh Display Dialog Box* (p. 1635) opens when **Draw Mesh** is selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update from Plane Tool
updates the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** from the plane tool orientation. The **Width**, **Height**, and **Depth** will also be updated. See *Using the Plane Tool* (p. 1367) for information about using the plane tool.

Auxiliary Fluid Inlet Direction
specifies the direction in which the auxiliary fluid enters the heat exchanger. See *Figure 15.6* (p. 777).

Pass-to-Pass Direction
specifies the direction in which the auxiliary fluid moves at the end of each pass through the heat exchanger. See *Figure 15.6* (p. 777).

Auxiliary Fluid
contains inputs to specify the properties of the auxiliary fluid.

Properties Method
specifies the method to specify the auxiliary fluid properties. You can choose from **constant-specific-heat** and **user-defined-enthalpy**.

Specific Heat
sets the specific heat of the auxiliary fluid (c_p in Equation 6–16 you choose **constant-specific-heat**.

Enthalpy UDF
sets the enthalpy as defined by the UDF selected from the drop-down list.

Auxiliary Fluid Flow Rate
sets the flow rate of the auxiliary fluid ($\dot{m}$ in Equation 6–16 in the Theory Guide).

Initial Temperature
sets an initial guess for the inlet temperature (T_{in} in Equation 6–11 and Equation 6–16 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Inlet Pressure
sets the auxiliary fluid inlet pressure. This value is specified only if **user-defined-enthalpy** is selected.

Inlet Quality
specifies the value of x in Equation 6–20 in the Theory Guide.

Pressure Drop
specifies the value of p_{in} in Equation 6–21 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Supplementary Auxiliary Fluid Stream
specifies properties of the supplementary auxiliary stream.

Supplementary Mass Flow Rate
specifies the supplementary fluid flow rate as **constant**, **polynomial** or **piecewise-linear**.

Supplementary Flow Temperature
specifies the supplementary fluid temperature as **constant**, **polynomial** or **piecewise-linear**.

Supplementary Flow Quality
specifies the supplementary fluid quality.

Create
saves all the settings in the dialog box.

Delete
deletes the group that is selected in the **HX Groups** list.

Replace
changes the parameters of the already existing group that is selected in the **HX Groups** list.

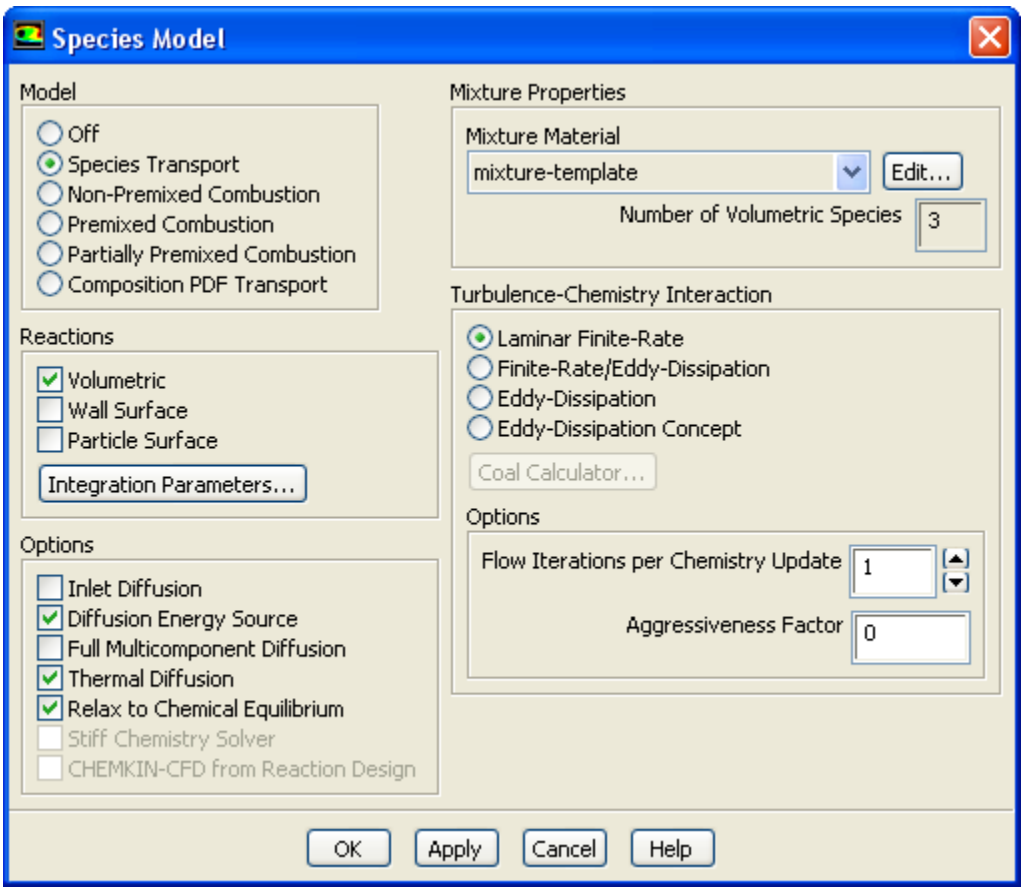
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set...

opens the *Ungrouped Macro Heat Exchanger Dialog Box* (p. 1672), in which you can define a new heat exchanger core model or read one from an external file.

35.3.18. Species Model Dialog Box

The **Species Model** dialog box allows you to set parameters related to the calculation of species transport and combustion. See *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801), *User Inputs for Wall Surface Reactions* (p. 829), *User Inputs for Particle Surface Reactions* (p. 835), *Using the Premixed Combustion Model* (p. 894), *Using the Partially Premixed Combustion Model* (p. 905), and *Steps for Using the Composition PDF Transport Model* (p. 911) for details about the items below.



Controls

Model

indicates which model, if any, is used to calculate species transport/combustion.

Off

disables species calculations.

Species Transport

enables the calculation of multi-species transport (either non-reacting or reacting, depending on the selection for **Reactions**). See *Modeling Species Transport and Finite-Rate Chemistry* (p. 799) for details.

Non-Premixed Combustion

enables the calculation of turbulent reacting flow using the non-premixed combustion model. See *Modeling Non-Premixed Combustion* (p. 839) for details. This option is available only for turbulent flows using the pressure-based solver.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Premixed Combustion
enables the premixed turbulent combustion model. See *Modeling Premixed Combustion* (p. 893) for details. This option is available only for turbulent flows using the pressure-based solver.

Partially Premixed Combustion
enables the partially premixed turbulent combustion model. See *Modeling Partially Premixed Combustion* (p. 905) for details. This option is available only for turbulent flows using the pressure-based solver.

Composition PDF Transport
enables the composition PDF transport model. See *Modeling a Composition PDF Transport Problem* (p. 911) for details. This option is available only for turbulent flows using the pressure-based solver.

Reactions
contains options related to the modeling of reacting flow. (This section of the dialog box appears only when **Species Transport** or **Composition PDF Transport** is the specified **Model**.)

Volumetric
enables the calculation of reacting flow using the finite-rate formulation. See *Volumetric Reactions* (p. 799) for details.

Wall Surface
enables the calculation of wall surface reactions. See *Wall Surface Reactions and Chemical Vapor Deposition* (p. 829) for details. This item will appear only if **Volumetric** is enabled.

Particle Surface
enables the calculation of particle surface reactions. See *Particle Surface Reactions* (p. 835) for details. This item will appear only if **Volumetric** is enabled.

Integration Parameters...
is a command button that opens the *Integration Parameters Dialog Box* (p. 1694). This button appears for the species transport model, when **Volumetric** is enabled under **Reactions** and **Stiff Chemistry Solver** is enabled under **Options** or when **Eddy-Dissipation Concept** is enabled under **Turbulence-Chemistry Interaction**.

Wall Surface Reaction Options
contains additional options for wall surface reactions. This portion of the dialog box appears only if **Wall Surface** is enabled under **Reactions**.

Heat of Surface Reactions
(if enabled) includes the heat release due to surface reactions in the energy equation. You must remember to set appropriate formation enthalpies (standard state enthalpies) if you enable this option.

Mass Deposition Source
(if enabled) includes the effect of surface mass transfer in the continuity equation.

Aggressiveness Factor
is a numerical factor which controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 (the default) is the most robust, but results in the slowest convergence.

Options
contains additional options for the **Species Transport** model and for the **Composition PDF Transport**. (This section will not appear in the dialog box for the other models.)

Inlet Diffusion
includes the diffusion flux of species at inlet.

Diffusion Energy Source
(if enabled) includes the effect of enthalpy transport due to species diffusion in the energy equation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Full Multicomponent Diffusion
enables the full multicomponent diffusion model. See *Full Multicomponent Diffusion* (p. 440) for details.

Thermal Diffusion
enables the thermal diffusion model. See *Thermal Diffusion Coefficients* (p. 442) for details.

Relax to Chemical Equilibrium
enables the calculation of the characteristic time for the turbulence-chemistry interaction models. See The Relaxation to Chemical Equilibrium Model for details.

Stiff Chemistry Solver
enables the calculations for modeling stiff laminar flames. See *Solution of Stiff Laminar Chemistry Systems* (p. 824) for details.

Liquid Micro-Mixing
is used to model liquid reactions. When the **Liquid Micro-Mixing** model is invoked, ANSYS FLUENT uses the **volume-weighted-mixing-law** formula to calculate the density.

CHEMKIN-CFD From Reaction Design
enables the use of reaction rates-of-production from Reaction Design's CHEMKIN module, coupled to ANSYS FLUENT's ISAT algorithm.

Thickened Flame Model
enables the modeling of laminar flames. This application is typically used as an LES combustion model for turbulent premixed and partially-premixed flames.

Include Temperature Fluctuations
enables the calculation of the multi-mode energy equation. This option is available when the composition PDF transport model is selected.

Mixture Properties
contains controls and information about the mixture being modeled. This section of the dialog box will not appear if **Premixed Combustion** is the selected under **Model**.

Mixture Material
contains a drop-down list of available mixture materials. When you first enable the **Species Transport** model, you can choose from all of the mixture materials defined in the database, or you can choose a "template" and define your own material. (Click **View...** to open the *FLUENT Database Materials Dialog Box* (p. 1744) and check the properties of the mixture material selected in the list.) See *Enabling Species Transport and Reactions and Choosing the Mixture Material* (p. 801) for details.

When you use the **Non-Premixed Combustion** or **Partially Premixed Combustion** model, this list will be inactive. The mixture material for a non-premixed or partially premixed combustion calculation will be determined from the content of the PDF file generated in ANSYS FLUENT using the **PDF Options** parameters.

Number of Volumetric Species
displays the number of gas-phase species in the selected **Mixture Material**. This is an informational display only; you cannot edit this value.

Number of Solid Species
displays the number of solid species defined in the selected **Mixture Material**. This is an informational display only; you cannot edit this value. (This list will appear only for **Species Transport** models involving **Wall Surface** reactions.)

Number of Site Species
displays the number of site species defined in the selected **Mixture Material**. This is an informational display only; you cannot edit this value. (This list will appear only for **Species Transport** models involving **Wall Surface** reactions.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence-Chemistry Interaction

indicates which model is to be used for turbulence-chemistry interaction when the **Species Transport** model with **Volumetric** reactions is used.

Laminar Finite-Rate

computes only the Arrhenius rate (see Equation 7–8 in the Theory Guide) and neglects turbulence-chemistry interaction.

Finite-Rate/Eddy-Dissipation

(for turbulent flows) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

Eddy-Dissipation

(for turbulent flows) computes only the mixing rate (see Equation 7–24 and Equation 7–25 in the Theory Guide).

Eddy-Dissipation Concept

(for turbulent flows) models turbulence-chemistry interaction with detailed chemical mechanisms (see Equation 7–24 and Equation 7–25 in the Theory Guide).

Coal Calculator...

opens the *Coal Calculator Dialog Box* (p. 1693).

Options

contains parameters related to the Laminar Finite-Rate or the Eddy-Dissipation Concept model. This section of the dialog box will appear when **Laminar Finite-Rate** or the **Eddy-Dissipation Concept** is selected for **Turbulence-Chemistry Interaction**.

Flow Iterations Per Chemistry Update

specifies how often ANSYS FLUENT will update the chemistry during the calculation. Increasing the number can reduce the computational expense of the chemistry calculations.

Aggressiveness Factor

is a numerical factor which controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 (the default) is the most robust, but results in the slowest convergence.

Volume Fraction Constant

specifies the value of C_ξ in Equation 7–27 in the Theory Guide.

Time Scale Constant

specifies the value of C_τ in Equation 7–28 in the Theory Guide.

Number of Grid Points in Flame

by default are 8 grid points.

PDF Options

contains options related for the non-premixed combustion model. (This section will appear only if **Non-Premixed Combustion** or **Partially-Premixed Combustion** is the selected **Model**.)

Inlet Diffusion

includes the diffusion flux of species at inlet.

Compressibility Effects

can be enabled to account for cases where substantial pressure changes occur in time and/or space when modeling a non-adiabatic system. See *Specifying the Operating Pressure for the System* (p. 845) for details.

Liquid Micro-Mixing

is used to model liquid reactions. When the **Liquid Micro-Mixing** model is invoked, ANSYS FLUENT uses the **volume-weighted-mixing-law** formula to calculate the density.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chemistry

tab contains the parameters to define problems using the chemistry model. See *Setting Up the Equilibrium Chemistry Model* (p. 843) for details.

State Relation

Equilibrium

enables the equilibrium chemistry model. See *Setting Up the Equilibrium Chemistry Model* (p. 843) for details.

Steady Flamelet

enables the steady laminar flamelet model. See The Laminar Flamelet Models Theory in the Theory Guide for details.

Unsteady Flamelet

enables the Eulerian unsteady laminar flamelet model.

Diesel Unsteady Flamelet

enables the diesel unsteady laminar flamelet model. See *Using the Diesel Unsteady Laminar Flamelet Model* (p. 850) for details.

Energy Treatment

Adiabatic

enables adiabatic modeling options for the problem.

Non-Adiabatic

enables nonadiabatic modeling options for the problem. See Non-Adiabatic Extensions of the Non-Premixed Model in the Theory Guide for details.

Coal Calculator

opens the *Coal Calculator Dialog Box* (p. 1693).

Stream Options

contains the parameters for the equilibrium chemistry model or the steady laminar flamelet model.

Secondary Stream

includes the secondary inlet stream in the model.

Empirical Fuel Stream

enables parameters to define fuel stream empirically. This option is available only with the full equilibrium chemistry model.

Empirical Secondary Stream

enables parameters to define secondary stream empirically. This option is available only with the full equilibrium chemistry model.

Model Settings

contains a list of parameter settings.

Operating Pressure

specifies the system operating pressure used to calculate the density using the ideal gas law. See *Specifying the Operating Pressure for the System* (p. 845) for details.

Fuel Stream Rich Flamability Limit

specifies the rich flammability limit for fuel stream when the equilibrium chemistry option is used. You will not set these if you have used the empirical definition option for fuel composition. See *Enabling the Rich Flammability Limit (RFL) Option* (p. 847) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Secondary Stream Flamability Limit
specifies the rich flammability limit for secondary stream when the equilibrium chemistry option is used. You will not set these if you have used the empirical definition option for fuel composition. See *Enabling the Rich Flammability Limit (RFL) Option* (p. 847) for details.

Empirical Fuel Lower Calorific Value
specifies the lower calorific value of fuel stream.

Empirical Fuel Specific Heat
specifies the specific heat value of fuel stream.

Empirical Fuel Molecular Weight
specifies the molecular weight of the fuel stream.

Empirical Secondary Lower Calorific Value
specifies the lower calorific value of secondary stream.

Empirical Secondary Specific Heat
specifies the specific heat value of secondary stream.

Empirical Secondary Molecular Weight
specifies the molecular weight of the secondary stream.

Options
contains options related to the steady flamelet model.

Create Flamelet
enables the **Import CHEMKIN Mechanism...** button that opens the *CHEMKIN Mechanism Import Dialog Box* (p. 2055) where you can import the **CHEMKIN** mechanism and thermodynamic data, to create a flamelet file. This option is available for the steady flamelet model. See *Setting Up the Steady and Unsteady Laminar Flamelet Models* (p. 847) for details.

Import Flamelet
enables the **Import Flamelet File...** button that opens the *The **Select File** Dialog Box* (p. 33) where you can select the existing flamelet in ANSYS FLUENT. You can also set the file type parameters to import the existing flamelet in ANSYS FLUENT. See *Setting Up the Steady and Unsteady Laminar Flamelet Models* (p. 847) for details. This option is available for the steady flamelet model.

File Type
contains the toggle buttons for two flamelet file types.

Standard
enables the import of an ASCII format standard flamelet file.

Oppdif
enables the import of a binary format OPPDIF flamelet file.

CFX-RIF
enables the import of an ASCII format CFX-RIF flamelet file.

Mixture Fraction Method
contains the three methods of computing the mixture fraction profile along the laminar flamelet.

Drake
calculates the mixture fraction using carbon and hydrogen elements.

Bilger
calculates the mixture fraction using hydrocarbon formula.

Nitrogen
calculates the mixture fraction in terms of nitrogen species.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Oppdif Flamelet Type
gives you have a choice of importing **Single** or **Multiple** OPPDIF files.

Flamelet Property File Name
opens the *The **Select File** Dialog Box* (p. 33) in which you can save the existing flamelet in ANSYS FLUENT to use when running an existing case.

Thermodynamic Database File Name
specifies a path for the thermodynamic database file to be read.

Boundary
tab contains the list of boundary species and related parameters. This is available only for equilibrium chemistry model. See *Defining the Stream Compositions* (p. 851) for details.

Species
contains the list of the species used in the problem.

Fuel
specifies the fuel species.

Oxid
specifies the oxidizing species.

Second
specifies the secondary species.

Boundary Species
allows to specify the species you want to add or remove from the model. You can type the species chemical formula in the text box below it.

Add
adds the species in the model.

Remove
removes the species from the model.

List Available Species
prints a list of all species in the thermodynamic database file (`thermo.db`) in the console window.

Temperature
specifies the temperature of different streams that you have defined.

Fuel
is the temperature of the fuel inlet in the model.

Oxid
is the temperature of the oxidizer inlet in the model.

Second
is the temperature of the secondary stream inlet in the model.

Specify Species in
allows to define the unit of species concentration.

Mass Fraction
allows to specify the species in terms of mass fraction.

Mole Fraction
allows to specify the species in terms of mole fraction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Control

tab contains the parameters for exclusion and inclusion of equilibrium species. This is available only for equilibrium chemistry model. See *Forcing the Exclusion and Inclusion of Equilibrium Species* (p. 862) for details.

Species Excluded from Equilibrium

lists the species excluded from equilibrium calculation.

Species

lists the slow-forming species that are zeroed in the initial flamelet profile.

Add

allows to add equilibrium species.

Remove

allows to remove equilibrium species.

List Available Species

prints a list of all species in the thermodynamic database file in the console window.

Flamelet Controls

allows you to adjust the controls for the flamelet solution. Note that the **Create Flamelet** option in the **Chemistry** tab must be selected for the **Steady Flamelet** model for these controls to be available.

Initial Fourier Number

sets the first time step for the solution.

Fourier Number Multiplier

increases the time step at subsequent times. Every time step after the first is multiplied by this value.

Relative Error Tolerance and Absolute Error Tolerance

specifies the local error controls during numerical integration.

Flamelet Convergence Tolerance

specifies the maximum absolute change in species fraction or temperature at any discrete mixture-fraction.

Maximum Integration Time

specifies the maximum total elapsed time for flamelet calculation. ANSYS FLUENT will stop the flamelet calculation after the total elapsed time has exceeded this value.

Flamelet

tab allows you to adjust the controls for the flamelet solution. See *Defining the Flamelet Controls* (p. 863) for details.

Flamelet Parameters

consist of the controls for the flamelet solution.

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f = 0$) and the fuel ($f = 1$).

Maximum Number of Flamelets

specifies the maximum number of laminar flamelet profiles to be calculated.

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated.

Unsteady Flamelet Parameters

consist of the controls for the unsteady flamelet solution.

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f = 0$) and the fuel ($f = 1$).

Mixture Fraction Lower Limit for Initial Probability

is the limit at which the unsteady flamelet model temporally convects and diffuses a marker probability equation through a steady-state ANSYS FLUENT flow-field.

Maximum Scalar Dissipation

is where flamelets extinguish at large scalar dissipation (mixing) rates.

Courant Number

is the number at which ANSYS FLUENT automatically selects the time step for the probability equation based on this convective Courant number.

Calculate Flamelets

begins the laminar flamelet calculation.

Display Flamelets...

opens the *Flamelet 3D Surfaces Dialog Box* (p. 1697) from which you can display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation.

Initialize Unsteady Flamelet Probability

initializes the unsteady flamelet and its probability marker equation.

Display Unsteady Flamelet...

opens the *Flamelet 2D Curves Dialog Box* (p. 1699) from which you can display 2D plots of the different variables.

Table

tab contains parameters to create the look-up table. See *Calculating the Look-Up Tables* (p. 871) for details of the items listed below.

Table Parameters

consist of the controls for the lookup table.

Number of Mean Mixture Fraction Points

is the number of discrete values of $\overline{f}$ at which the look-up tables will be computed.

Number of Secondary Mixture Fraction Points

is the number of discrete values of p_{sec} at which the look-up tables will be computed. This option is available only when a secondary stream has been defined.

Number of Mixture Fraction Variance Points

is the number of discrete values of $\overline{f'^2}$ at which the look-up tables will be computed. This option is available only when no secondary stream has been defined.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Mean Enthalpy Points
is the number of discrete values of enthalpy at which the three-dimensional look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system.

Minimum Temperature
is used to determine the lowest temperature for which the look-up tables are generated (see Figure 8.10: "Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems" in the Theory Guide). This option is available only if you are modeling a non-adiabatic system.

Include Equilibrium Flamelet
specifies that an equilibrium flamelet (i.e., $\chi = 0$) will be generated in ANSYS FLUENT and appended to the flamelet library before the PDF table is calculated. This option is available only when you are generating more than one laminar flamelet.

Calculate PDF Table
generates the look-up table.

Display PDF Table
opens the *PDF Table Dialog Box* (p. 2145) where you can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy.

Premixed
tab contains parameters needed to modify the piecewise-linear points. See *Modifying the Unburnt Mixture Property Polynomials* (p. 907) for the details.

Partially Premixed Mixture Properties
contains the list of properties that you can modify. **Edit...** opens the **Quadratic Mixture Fraction** dialog box where you can modify the values of the polynomial coefficients.

For each polynomial function of $\overline{f}$ under **Partially Premixed Mixture Properties** you can click **Edit...** and specify values for **Coefficient 1**, **Coefficient 2**, **Coefficient 3**, and **Coefficient 4** in the appropriate **Quadratic of Mixture Fraction** dialog box.

Premixed Combustion Model Options
contains options for the premixed combustion model. (This section will appear only if **Premixed Combustion** is the selected **Model**.)

Adiabatic
enables the adiabatic premixed combustion model, which calculates temperature using Equation 9–64 in the Theory Guide.

Non-Adiabatic
enables the non-adiabatic premixed combustion model, which calculates temperature using Equation 9–65 in the Theory Guide.

Premixed Model
contains options for choosing a premixed model.

C Equation
allows you to choose the C equation as described in C-Equation Model Theory.

Extended Coherent Flamelet Model
allows you to choose the Extended Coherent Flamelet model as described in Extended Coherent Flamelet Model Theory.

G Equation
allows you to choose the G equation as described in G-Equation Model Theory.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Flame Speed Model

contains the **Flame Speed Model** drop-down list, which allows you to choose between the **zimont** or the **peters** model constants for the Zimont premixed combustion model. (This section will appear only if **Premixed Combustion** or **Partially Premixed Combustion** is the selected **Model** and if the **C Equation** or **G Equation** premixed model is chosen.)

Turbulent Length Scale Constant

specifies the value of C_D in Equation 9–11 in the Theory Guide.

Turbulent Flame Speed Constant

specifies the value of A in Equation 9–9 in the Theory Guide.

Stretch Factor Coefficient

specifies the value of μ_{str} in Equation 9–16 in the Theory Guide.

Turbulent Schmidt Number

specifies the value of Sc_t in Equation 9–1 in the Theory Guide.

Wall Damping Coefficient

specifies the value of α_w in Equation 9–19.

Ewald Corrector

is enabled by default and described in Peters Flame Speed Model.

G Equation Settings

allows you to select either the **transport equation** or **algebraic** option for the calculation of the flame distance variance. Consult Peters Flame Speed Model in the Theory guide for the variance transport and algebraic equation expressions (Equation 9–6 and Equation 9–7).

Flame Curvature Source

includes the curvature source term in the G-Equation, which is the last term in Equation 9–4.

Extended Coherent Flamelet Model Constants

contains model constants for the Extended Coherent Flame Model. (This section will appear only if **Premixed Combustion** or **Partially Premixed Combustion** is the selected **Model** and if the **Extended Coherent Flame Model** flame speed model is chosen.) See *Modifying the Constants for the ECFM Flame Speed Closure* (p. 900) for details.

ITNFS Treatment

contains a drop-down list of the available ITNFS treatments: **constant-delta**, **meneveau**, **blint**, **poinsot**, and **constant**.

ITNFS Flame Thickness

sets the flame thickness (Equation 9–28 in the Theory Guide).

Turbulent Schmidt Number

set the turbulent Schmidt number (Sc_t).

Wall Flux Coefficient

set the wall flux coefficient.

PDF Transport Options

contains options for the Composition PDF Transport combustion model. (This section will appear only if **Composition PDF Transport** is the selected **Model**.)

Lagrangian

solves the composition PDF transport equation by stochastically tracking Lagrangian particles through the domain.

Eulerian

assumes a shape for the PDF, allowing Eulerian transport equations to be derived.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixing

tab contains the mixing models.

Mixing Model

contains options to specify the method for modeling molecular diffusion. (This section will appear only if **Composition PDF Transport** is the selected **Model**.) See Particle Mixing in the Theory Guide for details.

Modified Curl

enables the modified curl model for molecular diffusion.

IEM

enables the IEM model for molecular diffusion.

EMST

enables the EMST mixing model for molecular diffusion.

Mixing Constant

specifies the value of the mixing constant C_ϕ in Equation 11–6 and Equation 11–8 in the Theory Guide.

Boundary

tab allows you to define the fuel and oxidizer compositions. This is only available if you select **Eulerian** as the **PDF Transport Option**.

Species

consists of the fuel species and the oxidizer.

Fuel

is the mole or mass fraction of the fuel stream. The sum of mass or mole fractions of all species in the fuel stream should be 1.

Oxidizer

is the mole or mass fraction of the oxidizer stream. The sum of mass or mole fractions of all species in the fuel oxidizer stream should be 1.

Specify Species in

specifies the species as a **Mass Fraction** or **Mole Fraction**.

Control

tab contains Lagrangian PDF transport parameters.

PDF Transport Parameters

allows you to set the **Particles Per Cell**.

Particles Per Cell

sets the number of PDF particles per cell. Higher values of this parameter will reduce statistical error, but increase computational time.

Local Time Stepping

toggles the calculation of local time steps. If this option is disabled, then you will need to specify the **Time Step** directly (see Equation 11–4 in the Theory Guide). This option is available for steady-state simulations.

Convection #

specifies the particle convection number (see Δt_{conv} in Equation 11–4 in the Theory Guide).

Diffusion #

specifies the particle diffusion number (see Δt_{diff} in Equation 11–4 in the Theory Guide).

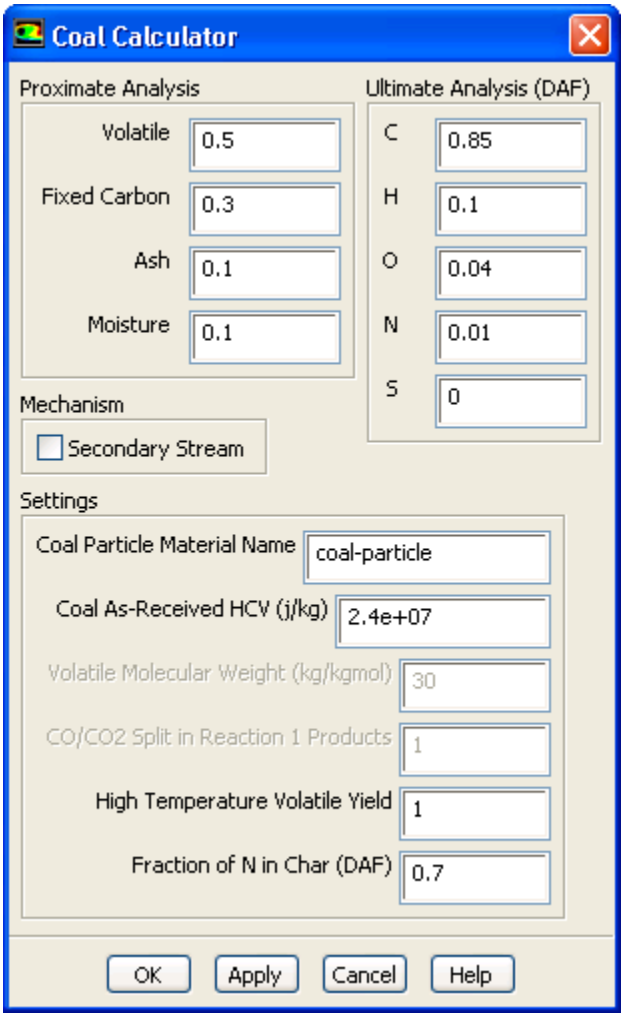
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixing #

specifies the particle mixing number (see Δt_{mix} in Equation 11–4 in the Theory Guide).

35.3.19. Coal Calculator Dialog Box

The **Coal Calculator** dialog box automates the calculations described in *Additional Coal Modeling Inputs in ANSYS FLUENT* (p. 858).



Proximate Analysis

is the mass fraction of **Volatile**, **Fixed Carbon**, **Ash**, and **Moisture** in the coal.

Volatile

is the fraction of the volatile component.

Fixed Carbon

is calculated as one minus the sum of the actual **Volatile**, **Ash**, and **Moisture** fractions.

Ash

is the fraction of ash.

Moisture

is the moisture fraction in the coal.

Ultimate Analysis (DAF)

is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mechanism

allows you to set the mechanisms.

Secondary Stream

when enabled, allows you to set the two mixture fraction model with the primary stream representing char as $C<\mathrm{s}>$, and an empirical secondary stream representing the volatiles. This is available when using the non-premixed combustion model.

One-step Reaction

is defined in *Equation 16–6* (p. 821).

Two-step Reaction

involves oxidation of volatiles to CO in the first reaction and oxidation of CO to CO_2 in the second reaction, as described in *Equation 16–7* (p. 821).

Include SO2

when enabled, allows you to input the atomic mass fraction of sulphur, **S**, which appears under **Ultimate Analysis**.

Options

Wet Combustion

when enabled will activate the DPM **Wet Combustion** option by default in all injections created after the **OK** button is clicked in the **Coal Calculator** dialog box.

Settings

is where you will specify the values used in the calculation.

Coal Particle Material Name

is the name of the DPM combusting particle material. The default name is *coal-particle*.

Coal As-Received HCV

is the higher caloric value of the coal.

Volatile Molecular Weight

is the molecular weight of pure volatiles.

CO/CO2 Split in Reaction 1 Products

can be used to specify the molar fraction of CO to CO_2 in the first reaction of *Equation 16–7* (p. 821). The default value of 1 implies that all carbon is reacted to CO, with no CO_2 produced.

High Temperature Volatile Yield

is where the enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction.

Fraction of N in Char (DAF)

is used in calculating the split of atomic nitrogen for the Fuel NOx model.

Coal Dry Density

is used to calculate the **Volume Fraction** of liquid-water for the **Wet Combustion** option in the **Injections** dialog box.

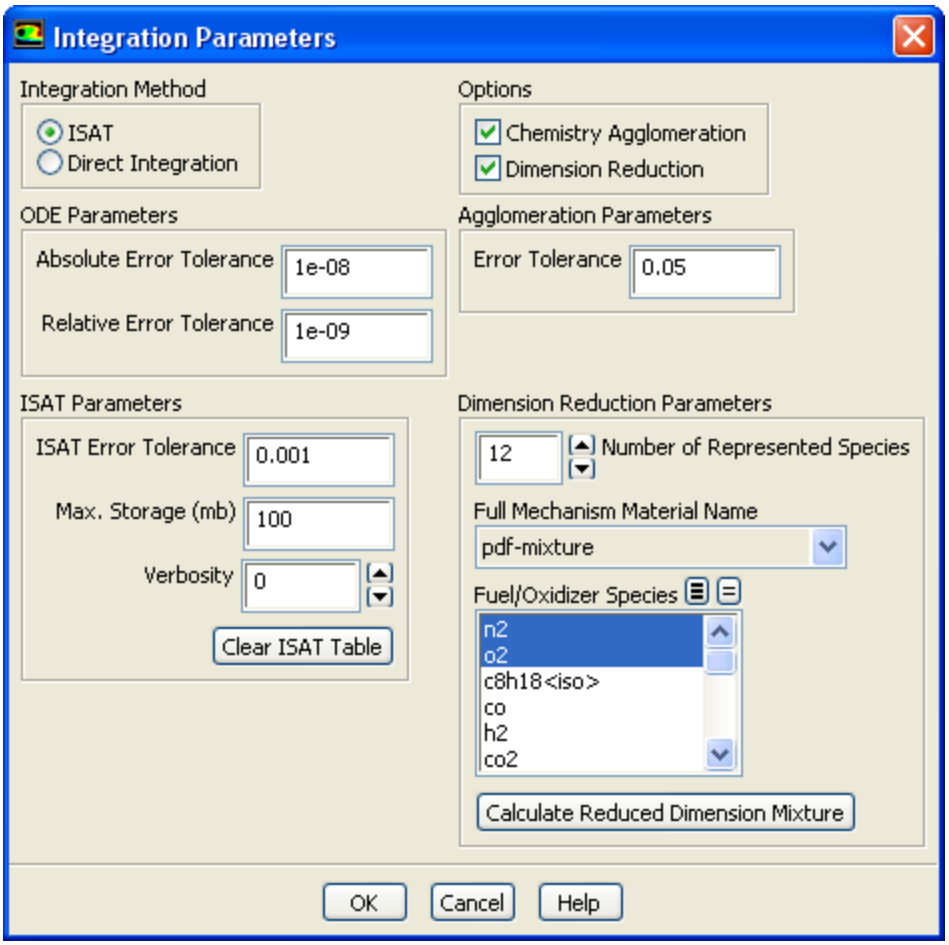
Gas Phase Reaction

lists the reaction based on your entries for the proximate and ultimate analyses.

35.3.20. Integration Parameters Dialog Box

The **Integration Parameters** dialog box allows you to set the parameters for the integration of the chemical source term S in Equation 11–10 in the Theory Guide. See *Using ISAT* (p. 924) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Integration Method

contains options to choose the method for integration.

ISAT

enables the ISAT option and expands the dialog box to include inputs for ISAT parameters.

Direct Integration

enables the direct integration method to integrate the chemical source term in the calculation.

ODE Parameters

contains options to specify the error tolerances.

Absolute Error Tolerance

specifies the absolute error tolerance.

Relative Error Tolerance

specifies the relative error tolerance.

ISAT parameters

contains inputs required for ISAT integration method.

ISAT Error Tolerance

controls the numerical error in ISAT liner interpolation. Decrease this value to get accurate minor species and pollutant predictions.

Max. Storage

is the maximum RAM used by the ISAT table, and has a default value is 100 MB.

Verbosity

specifies the level of detail at which you can monitor the ISAT performance.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clear ISAT Table
purges the ISAT table.

Options
contains options to choose the method for chemistry acceleration.

Chemistry Agglomeration
when enabled, provides additional run-time improvement, with a corresponding decrease in accuracy.

Dimension Reduction
is a chemistry acceleration method in addition to ISAT storage-retrieval and Cell Agglomeration, providing faster chemistry calculations with a corresponding loss of accuracy.

Agglomeration Parameters
allows you to specify the **Error Tolerance**. This determines the size of the clusters and by default the value is 0.05

Dimension Reduction Parameters
contains settings to accelerate the chemistry.

Number of Represented Species
must be greater than 10 and less than the number of species in the full mechanism. The **Number of Represented Species** must also be less than 50 minus the number of unrepresented elements (the number of chemical elements in the unrepresented species).

Full Mechanism Material Name
is typically the name of the CHEMKIN mechanism that you imported.

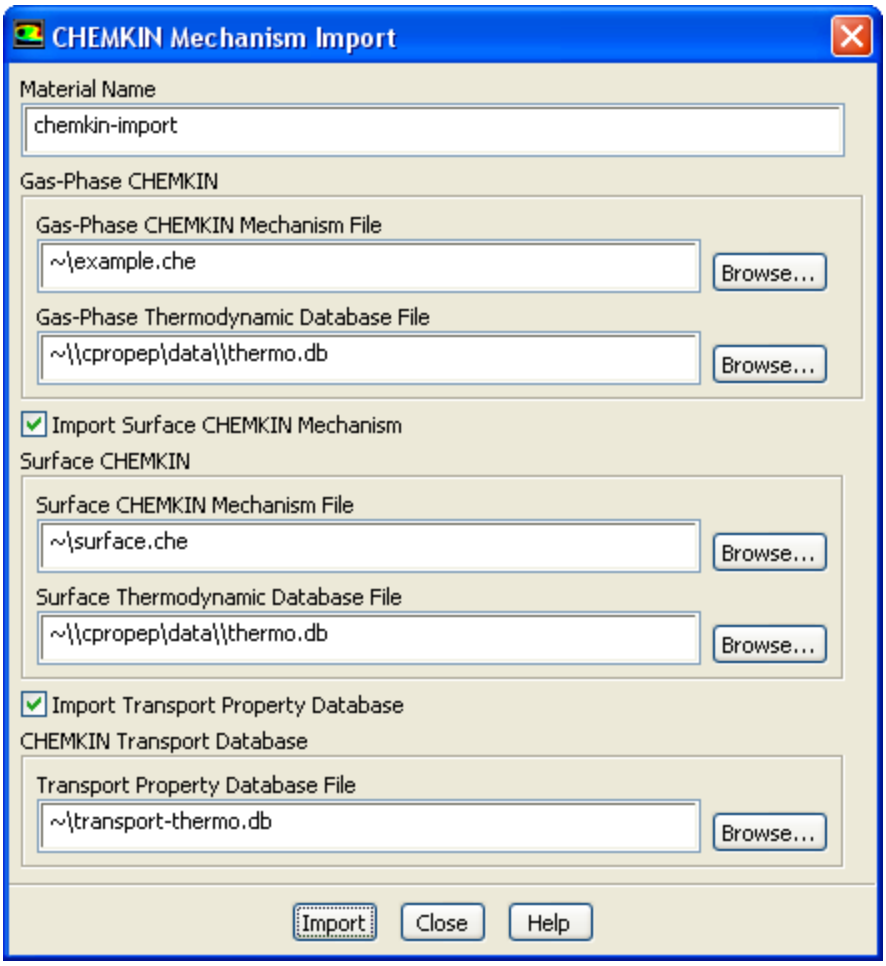
Fuel/Oxidizer Species
is where the boundary and initial fuel and oxidizer, as well as product species, are set as represented species.

Create Reduced Dimension Mixture
creates a new mixture material called `reduced-dimension-mixture`, which contains the represented species as well as proxy 'species' for the unrepresented elements.

35.3.21. Chemkin Mechanism Import Dialog Box

The **Chemkin Mechanism Import** dialog box allows you to read the chemical kinetic mechanism and thermodynamic data. See *Setting Up the Steady and Unsteady Laminar Flamelet Models* (p. 847) for details on the items listed below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Gas-Phase CHEMKIN

contains parameters to import the gas phase chemkin mechanism.

Gas-Phase CHEMKIN Mechanism File

specifies the path of the Chemkin file to be read.

Gas-Phase Thermodynamic Database File

specifies the location of the thermodynamic database.

Import Surface Chemkin Mechanism

enables parameters to import the surface chemkin mechanism.

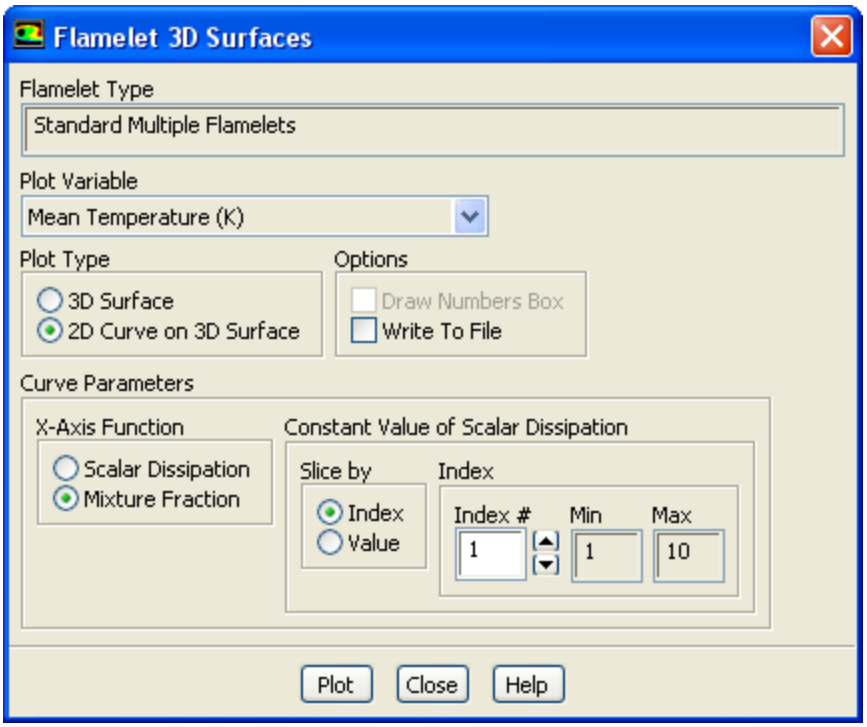
Import Transport Property Databases

(optional) allows to import transport properties.

35.3.22. Flamelet 3D Surfaces Dialog Box

The **Flamelet 3D Surfaces** dialog box allows you to display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation. See *Postprocessing the Flamelet Data* (p. 868) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Flamelet Type
indicates the type of flamelets whose surfaces will be displayed.

Plot Variable
enables you to choose temperature or species fraction as the variable to be plotted.

Plot Type
consists of options for plot type.

3D Surface
enables plotting on 3D surfaces.

2D curve on 3D surface
enables plotting of a 2D curve on a 3D surface.

Options
consists of the following parameters:

Draw Numbers Box
enables the display of a wireframe box with the numerical limits in each coordinate direction.

Write To File
enables saving the plot data to a file.

Curve Parameters
consists of controls related to plot display.

X-Axis Function
consists of the function against which the plot variable will be displayed.

Scalar Dissipation
enables display of plot variable against the scalar dissipation function.

Mixture Fraction
enables display of plot variable against the mixture fraction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constant Value of Scalar Dissipation

consists of the controls to specify the type of discretization (i.e., how the flamelet data will be sliced) for the variable that is being held constant.

Slice by

consists of the controls to specify discretization.

Index

enables you to specify discretization index of the variable that is being held constant.

Value

enables you to specify the numerical value of the variable that is being held constant.

Index

consists of the controls that are displayed when you enable **Index** under **Slice by**.

Index #

displays the index number.

Min

displays the minimum of the range of integer values that you are allowed to choose from.

Max

displays the maximum of the range of integer values that you are allowed to choose from.

Value

consists of the controls that are displayed when you enable **Value** under **Slice by**.

Value

enables you to enter the numerical value of the variable that is being held constant.

Min

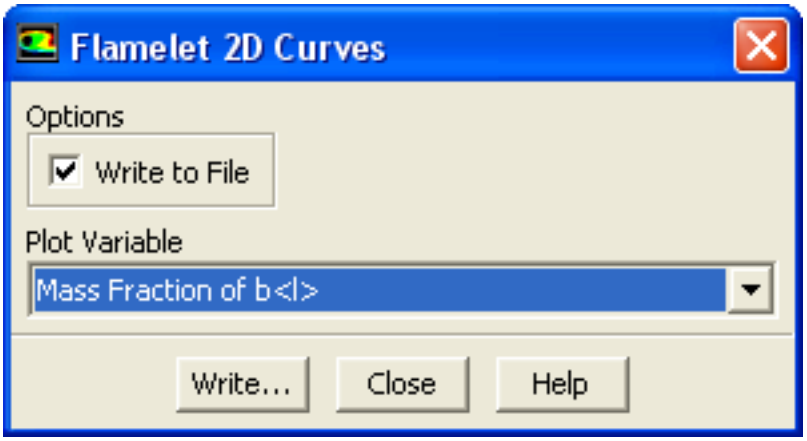
displays the minimum of the range of integer values that you are allowed to choose from.

Max

displays the minimum of the range of integer values that you are allowed to choose from.

35.3.23. Flamelet 2D Curves Dialog Box

The **Flamelet 2D Curves** dialog box allows you to display or write 2D curves of the unsteady flamelet. See *Postprocessing the Flamelet Data* (p. 868) for details.



Options

gives you the option to plot or **Write to File** 2D curves.

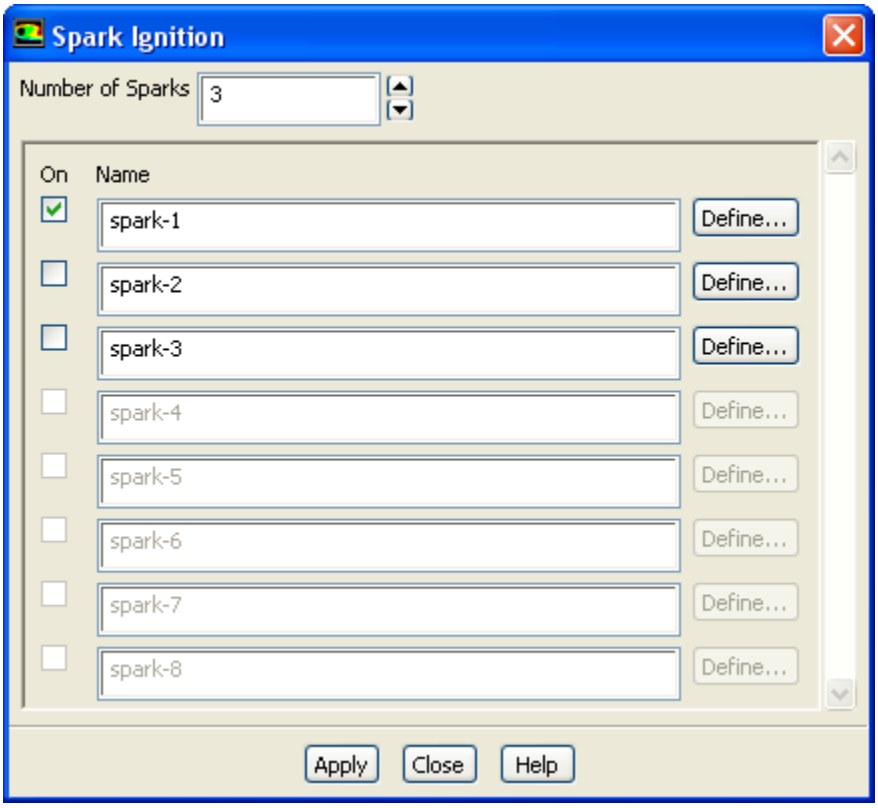
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Variable
consists of a drop-down list of variables that you can plot or write.

Plot
is available by default. When the **Write to File** option is enabled the **Plot** button changes to a **Write...**.

35.3.24. Spark Ignition Dialog Box

The **Spark Ignition** dialog box allows you to define multiple sparks (see *Spark Model* (p. 929) for details).



Controls

Number of Sparks
is the quantity of sparks you would like to include in your simulation. You can define up to 16 sparks.

On
if enabled, activates those sparks that will be included in the simulation.

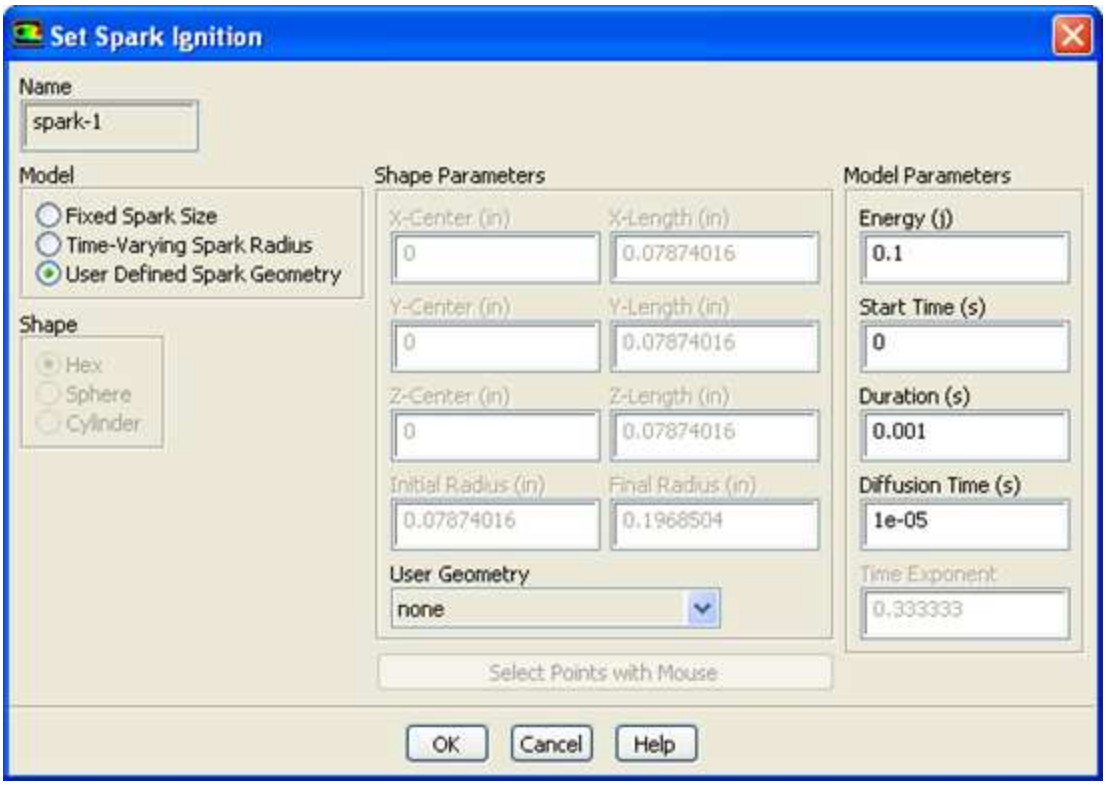
Name
is the name of the spark. You can specify a name, or use the default name.

Define...
opens the *Set Spark Ignition Dialog Box* (p. 1700).

35.3.25. Set Spark Ignition Dialog Box

The **Set Spark Ignition** dialog box allows you to set the parameters related to the spark ignition model (see *Spark Model* (p. 929) for details).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
displays the name of the spark being defined.

Model
contains the available spark ignition models.

Fixed Spark Size
enables the spark ignition calculations using the fixed spark size model.

Time-Varying Spark Radius
enables the spark ignition calculations using the time varying spark radius.

User Defined Spark Geometry
allows you to hook a user-defined function to define custom spark kernel volume shapes.

Shape
contains the different shapes of the spark. This option is available only for **Fixed Spark Size** model

Hex
enables hex shape spark.

Sphere
enables spherical shape spark.

Cylinder
enables cylindrical shape spark.

ECFM Spark Model
contains model variants used to define the value of the flame surface density. These models appear only when the **Extended Coherent Flamelet Model** (*Setting Up the Extended Coherent Flame Model* (p. 899)) is selected in the **Species Model** dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boudier
is available only as a time dependent kernel geometry. The geometric growth rate is the same as that used for the basic spark model with other combustion models. See Boudier Model.

Teraji
is available only as a time dependent kernel geometry. See Teraji Model.

Zimont
operates in the same way as the spark model used with the Zimont combustion model. The geometry of the spark may be time dependent or fixed. See Zimont Model.

Constant Value
allows you to specify the **Flame Surface Density**. See for details.Constant Value Model

User Defined Sigma Source
allows you to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel.

Shape Parameters
contains the parameters needed to define the shape and size of the spark.

X, Y, and Z-Center
specifies the points in x, y, and z direction.

X, Y, and Z-Length
specifies the length in x, y, and z direction.

Initial Radius
specifies the initial spark radius.

Final Radius
specifies the final spark radius.

User Geometry
allows you to hook a user-defined function to define custom spark kernel volume shapes. For details about this user-defined function, please refer to DEFINE_SPARK_GEOM in the UDF Manual.

Model Parameters
contains the parameters needed to define the spark.

Energy
contains the total energy input by the spark.

Start Time
is the time of spark ignition initialization.

Duration
is the duration of the spark ignition.

Diffusion Time
is the spark diffusion time in seconds.

Time Exponent
is the time exponent specified when the **Time-Varying Spark Radius** is used.

Flame Surface Density
is defined by Equation 13–5, when the extended coherent flamelet (ECFM) combustion model is used. The value of the flame surface density must be set within the spark region and can be specified when **Constant Value** is selected under **ECFM Spark Model**.

User Sigma Source
allows you to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel. For more detail about this user-defined function, please refer to Hooking

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

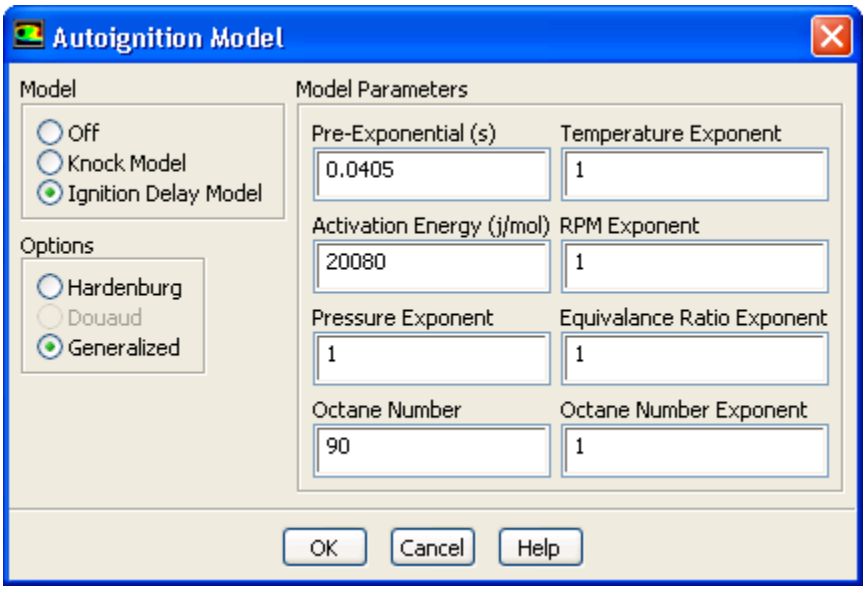
DEFINE_ECFM_SPARK_SOURCE UDFs of the UDF Manual. This selection is available when **User Defined Sigma Source** is enabled under **ECFM Spark Model**.

Select Points with Mouse

allows to select the shape parameters of the spherical spark using mouse.

35.3.26. Autoignition Model Dialog Box

The **Autoignition Model** dialog box allows you to set the parameters related to the **Knock Model** or the **Ignition Delay Model**. See *Modeling Engine Ignition* (p. 929) for details.



Controls

Model

contains options to disable or enable models.

Off

disables the model.

Knock Model

enables the knock model. With the **Premixed Combustion** or **Partially Premixed Combustion** models selected, only the **Knock Model** can be turned on.

Ignition Delay Model

enables the ignition delay model.

Options

contains two correlation options that exist with each model.

Douaud

option is used for knock in spark ignition engines. The modeling parameters that are specified in the GUI for this option are the **Pre Exponential**, **Pressure Exponent**, **Activation Temperature**, **Octane Number**, and **Octane Exponent** (Equation 13–12 in the Theory Guide).

Generalized

enables generalized correlation described by Equation 13–13 in the Theory Guide. It requires the same parameters as in the ignition delay model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hardenburg
enables Hardenburg correlation, which is used for heavy-duty diesel engines. This option is activated only for the ignition delay model.

Model Parameters
contains parameters related to the selected model. See *Using the Autoignition Models* (p. 933) for the details about the parameters in this dialog box.

Pre-Exponential
see Equation 13–13 and Equation 13–12 in the Theory Guide.

Activation Temperature
see Equation 13–12 in the Theory Guide.

Pressure Exponent
see Equation 13–13 and Equation 13–12 in the Theory Guide.

Octane Number
see Equation 13–13 and Equation 13–12 in the Theory Guide.

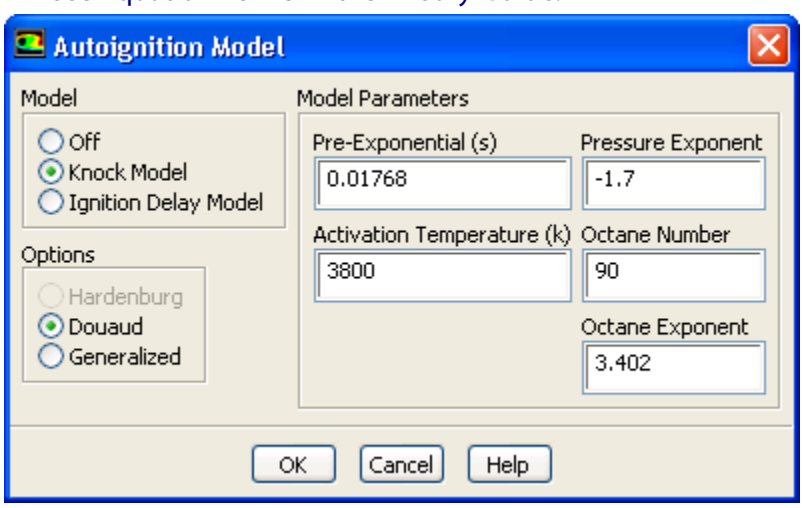
Octane Number Exponent
see Equation 13–13 and Equation 13–12 in the Theory Guide.

Activation Energy
see Equation 13–13 and Equation 13–15 in the Theory Guide.

Temperature Exponent
see Equation 13–13 in the Theory Guide.

RPM Exponent
see Equation 13–13 in the Theory Guide.

Equivalence Ratio Exponent
see Equation 13–13 in the Theory Guide.



Options
contains two correlation options that exist with this model.

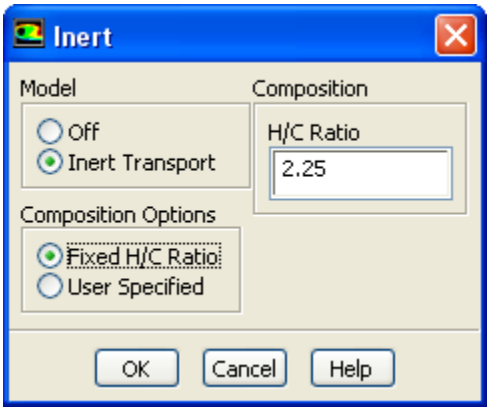
Douaud
enables Douaud correlation (Equation 13–12 in the Theory Guide) used for knock in SI engines.

Generalized
option (Equation 13–13 in the Theory Guide) in the knock model require the same parameters as in the ignition delay model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3.27. Inert Dialog Box

The **Inert** dialog box allows you to set the parameters related to the inert model. For details, see *Setting Up the Inert Model* (p. 880).



Controls

Model

allows you to enable or disable the inert model.

Off

disables the model.

Inert Transport

enables the inert model.

Composition Options

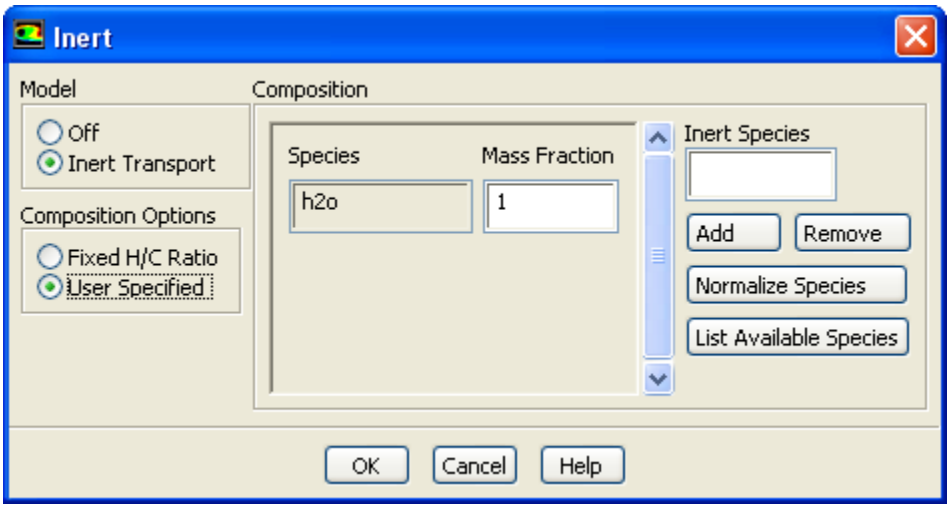
allows you to select a fixed H/C ratio or to specify the composition.

Fixed H/C Ratio

allows you to specify a fixed ratio of hydrogen to carbon in the **H/C Ratio** field.

User Specified

allows you to specify an arbitrary composition for the inert stream.



Composition

allows you to set the H/C ratio or the mass fraction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **Fixed H/C Ratio** is selected under **Composition Options**, the following option(s) are available:

H/C Ratio
specifies the fixed ratio of hydrogen to carbon when the **Fixed H/C Ratio** option is enabled.

When **User Specified** is selected under **Composition Options**, the following option(s) are available:

Species
lists the inert species name.

Mass Fraction
displays the mass fraction of the corresponding species.

Inert Species
allows you to specify the name of the inert species.

Add
adds the specified inert species to the species list.

Remove
removes the specified inert species from the species list.

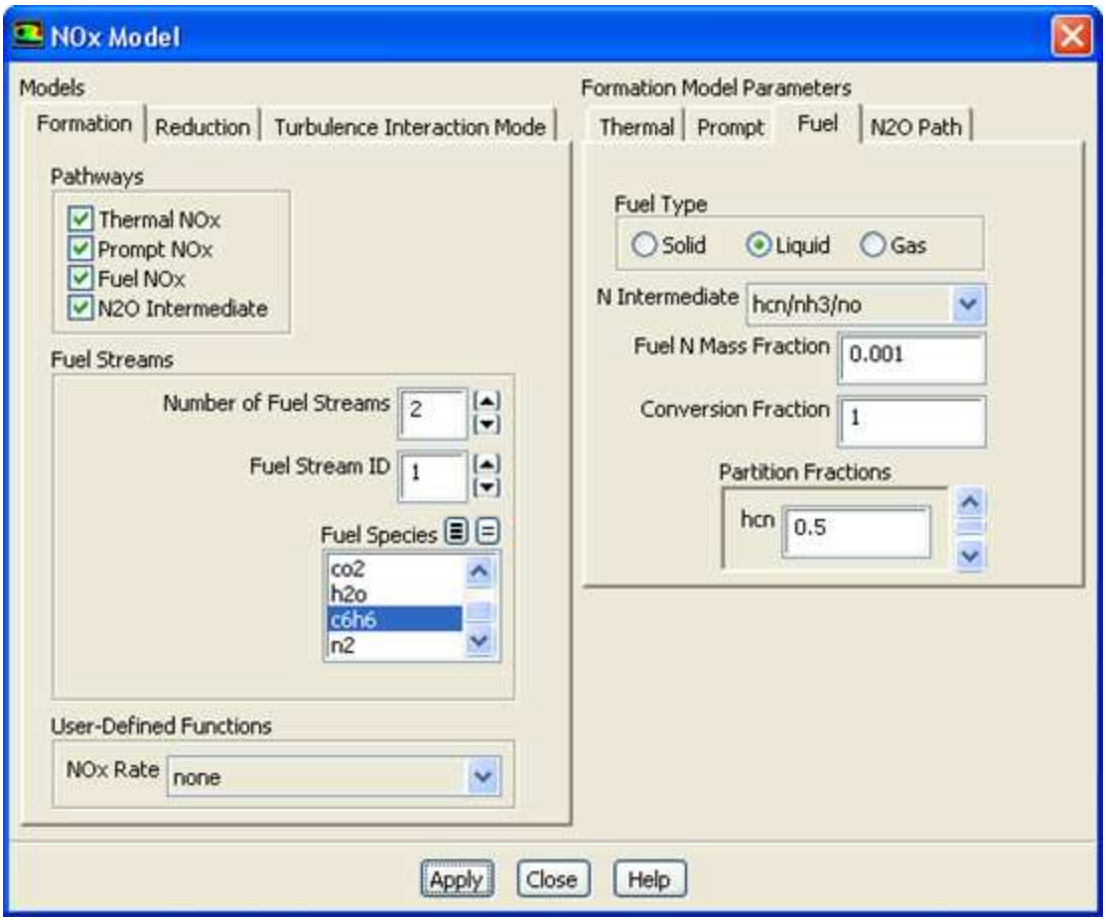
Normalize Species
makes sure the species mass fractions add up to 1.

List Available Species
lists all species in the thermodynamic database file (`thermo.db`) in the console window,

35.3.28. NOx Model Dialog Box

The **NOx Model** dialog box allows you to set parameters related to the NOx postprocessor. See *Using the NOx Model* (p. 943) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Models

contains tabs for defining the models used to calculate the NOx production.

Formation

contains the parameters to define the NOx model formation.

Pathways

contains toggle buttons for activating the NOx models to be used for the calculation of NO and HCN concentrations.

Thermal NOx

enables calculation of thermal NOx.

Prompt NOx

enables the calculation of prompt NOx.

Fuel NOx

enables the calculation of fuel NOx. When using the non-premixed combustion model, the **Fuel NOx** option is only available if the DPM model is also enabled.

N2O Intermediate

enables the formation of NOx through an N_2O intermediate. This option will only appear if one of the previously listed NOx models is enabled.

Fuel Streams

allows you to define multiple fuel streams for prompt NOx and fuel NOx formation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Fuel Streams
sets the number of fuel streams. You are allowed up to three fuel streams.

Fuel Stream ID
specifies the fuel stream you are defining in the **PDF Stream** drop-down list, the **Fuel Species** selection list, the **Prompt** tab, and the **Fuel** tab.

PDF Stream
specifies the PDF stream species associated with a particular **Fuel Stream ID**, when calculating fuel NOx formation in conjunction with the non-premixed combustion model. You can select either the **primary** or **secondary** fuel streams, as defined in the PDF table.

Fuel Species
is a list containing all of the defined species, which allows you to specify the species that is the fuel associated with a particular **Fuel Stream ID**. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10. Note that when the non-premixed combustion model is enabled, your selection in the **Fuel Species** list only applies to prompt NOx calculations.

User-Defined Functions
contains the **NOx Rate** drop-down list, which allows you to use a user-defined function (UDF) to contribute to the rate of NOx production. See the separate UDF Manual for details. Note that you may also use a UDF to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

Reduction
allows you to specify the reduction methods.

Methods
contains the list of reduction methods.

Reburn
enables the calculation of NOx reburning effects.

SNCR
enables the calculation of NOx reduction by the SNCR method.

Turbulence Interaction Mode
contains parameters related to the effect of turbulent fluctuations on the NOx formation. See NOx Formation in Turbulent Flows in the Theory Guide for details.

PDF Mode
is a drop-down list containing options that take into account turbulent fluctuations when you compute the specified NOx formation. See *Setting Turbulence Parameters* (p. 955) for details.

none
specifies the use of laminar NOx rate calculations, so that the effects of turbulence are ignored.

temperature
includes fluctuations of temperature.

temperature/species
includes fluctuations of the temperature and the mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction
includes fluctuations of the mixture fraction(s). This is available for non-premixed combustion calculations only.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF Type
allows you to specify the shape of the PDF.

beta
models the PDF using Equation 14–108 in the Theory Guide.

gaussian
models the PDF using Equation 14–111 in the Theory Guide.

PDF Points
controls the number of points at which the beta function in Equation 14–105 or Equation 14–106 in the Theory Guide will be integrated. The default value of 10, which indicates that the beta function will be integrated at 10 points on a histogram basis. The default value should yield an accurate solution with a reasonable computation time. Increasing this value may improve accuracy, but will also increase the computation time. This text box is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance
allows you to specify the form of the transport equation that is solved to calculate the temperature variance.

algebraic
is an approximate form of the transport equation (see Equation 14–114 in the Theory Guide).

transported
solves Equation 14–113 in the Theory Guide.

Tmax Option
provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax
sets the limit as the maximum temperature in the flow field.

local-tmax-factor
yieldscell-basedmaximumtemperaturelimitsby multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax
sets the limit for each cell to be the value entered in **Tmax**.

user-defined
allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a UDF and selected it in the **Formation** tab.

Species
is a drop-down list which appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the NOx rate calculations.

Formation Model Parameters
contains the tabs used to define the NOx pathways.

Thermal
contains parameters for modeling thermal NOx. (The contents of this tab will appear only if **Thermal NOx** is enabled in the **Formation** tab.)

[O] Model
is a drop-down list in which you can select the method to be used for calculation of thermal NOx. To choose the equilibrium method, select **equilibrium**. To choose the partial equilibrium

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

method, select **partial-equilibrium**. To choose the predicted O concentration method, select **instantaneous**. See Method 1: Equilibrium Approach, Method 2: Partial Equilibrium Approach, and Method 3: Predicted O Approach in the Theory Guide for details.

[OH] Model

is a drop-down list in which you can select the method to be used for calculation of thermal NOx. To exclude OH, select **none**. To choose the partial equilibrium method, select **partial-equilibrium**. To choose the predicted OH concentration method, select **instantaneous**. See Method 1: Exclusion of OH Approach, Method 2: Partial Equilibrium Approach, and Method 3: Predicted OH Approach in the Theory Guide for details.

UDF Rate

provides options for the treatment of the NOx production specified by the UDF selected in the **Formation** tab.

Replace FLUENT Rate

replaces ANSYS FLUENT's thermal NOx rate calculations with the custom NOx rate produced by your UDF.

Add to FLUENT Rate

adds the custom NOx rate produced by your UDF to ANSYS FLUENT's thermal NOx rate calculations.

Prompt

contains parameters for modeling prompt NOx. (The contents of this tab will appear only if **Prompt NOx** is enabled in the **Formation** tab.) The settings made in this tab will be associated with a particular fuel stream, specified in the **Fuel Stream ID** text box in the **Formation** tab.

Fuel Carbon Number

specifies the number of carbon atoms per fuel molecule.

Equivalence Ratio

is the ratio of the actual fuel/air ratio to the stoichiometric fuel/air ratio.

UDF Rate

provides options for the treatment of the NOx production specified by the UDF selected in the **Formation** tab.

Replace FLUENT Rate

replaces ANSYS FLUENT's prompt NOx rate calculations with the custom NOx rate produced by your UDF.

Add to FLUENT Rate

adds the custom NOx rate produced by your UDF to ANSYS FLUENT's prompt NOx rate calculations.

Fuel

contains parameters for modeling fuel NOx. (The contents of this tab will appear only if **Fuel NOx** is enabled in the **Formation** tab.) The settings made in this tab will be associated with a particular fuel stream, specified in the **Fuel Stream ID** text box in the **Formation** tab.

Fuel Type

specifies the type of fuel NOx to be calculated.

Solid

enables the calculation of solid fuel NOx.

Liquid

enables the calculation of liquid fuel NOx.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gas

enables the calculation of gas fuel NOx.

N Intermediate

allows you to specify any one of the **hcn**, **nh3**, or **hcn/nh3/no** as the intermediate species. See Fuel NOx Formation in the Theory Guide for details.

Volatile N Mass Fraction

specifies the mass fraction of nitrogen in the volatiles. This parameter appears only for **Solid** fuel NOx calculations.

Fuel N Mass Fraction

specifies the mass fraction of nitrogen in the fuel. This parameter appears only for **Gas** or **Liquid** fuel NOx calculations.

Conversion Fraction

specifies the overall mass fraction of fuel N (for gas and liquid fuels), or volatile N or char N (for solid fuels), that will be converted to intermediate species and/or product NO.

Partition Fractions

specifies the mass fraction of the converted fuel N (for gas and liquid fuels), or volatile N or char N (for solid fuels), that will become **hcn** and **nh3**. The fraction that will become NO will be calculated by the remainder. This option will appear only if you have selected **hcn/nh3/no** for the **N Intermediate** or **Char N Conversion** drop-down lists.

Char N Conversion

is a drop-down list in which you can select **no**, **hcn**, **nh3**, or **hcn/nh3/no** as the species to which the char N is converted (when you are calculating solid fuel NOx). This parameter appears only for **Solid** fuel NOx calculations. See *Setting Solid (Coal) Fuel NOx Parameters* (p. 950) for details.

Char N Mass Fraction

specifies the mass fraction of nitrogen in the char. This parameter appears only for **Solid** fuel NOx calculations.

BET Surface Area

sets the BET internal pore surface area (see BET Surface Area in the Theory Guide for details) of the particles. This parameter appears only for **Solid** fuel NOx calculations.

UDF Rate

provides options for the treatment of the NOx production specified by the UDF selected in the **Formation** tab.

Replace FLUENT Rate

replaces ANSYS FLUENT's fuel NOx rate calculations with the custom NOx rate produced by your UDF.

Add to FLUENT Rate

adds the custom NOx rate produced by your UDF to ANSYS FLUENT's fuel NOx rate calculations.

N2O Path

contains the method to be used for formation of NO through an N_2O intermediate. (The contents of this tab will appear only if **N20 Intermediate** is enabled in the **Formation** tab.)

N2O Model

contains the drop-down list of available N2O models.

quasi-steady

enables the quasi-steady-state method of calculation (The transport equation for the species N_2O will not be solved).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

transported-simple

enables the transported simple method of calculation (The pollutant species N_2O is added in the species list and it's mass fraction will be calculated using the transport equations).

UDF Rate

provides options for the treatment of the NOx production specified by the UDF selected in the **Formation** tab.

Replace ANSYS FLUENT Rate

replaces the NOx rate calculated by ANSYS FLUENT using N_2O intermediates with the custom NOx rate produced by your UDF.

Add to ANSYS FLUENT Rate

adds the custom NOx rate produced by your UDF to the NOx rate calculated by ANSYS FLUENT using N_2O intermediates.

Reduction Method Parameters

contains tabs that allow you to define the methods of reduction. (These tabs are do not appear unless a reduction method has been enabled in the **Reduction** tab.)

Reburn

allows you to define the NOx reduction when **Reburn** is enabled in the **Reduction** tab.

Reburn Model

contains the drop-down list of reburn methods.

instantaneous[CH]

activates instantaneous method in the **Reburn Model**. When you choose this method a warning to include CH, CH_2, and CH_3 will be displayed.

partial-equilibrium

activates partial method in the **Reburn Model**.

Reburn Fuel Species

contains reburn fuel species drop-down list.

Equivalent Fuel Type

contains equivalent fuel type drop-down list.

SNCR

allows you to define the NOx reduction when **SNCR** is enabled in the **Reduction** tab.

Injection Method

contains the parameters for NOx reduction by SNCR method.

gaseous

includes ammonia or urea as a gas-phase pollutant species from the injection locations.

liquid

includes ammonia or urea as a liquid-phase pollutant species from the injection locations.

Reagent Species

allows you to specify the reagent species as either ammonia (**nh3**) or urea (**co<nh2>2**)

Reagent Fraction in Stream

allows you to specify the mass fraction of the reagent in the reagent stream. The remaining mass fraction is assumed to be water. If you enabled a secondary stream in your PDF calculation, by default the secondary stream will act as the reagent stream. Note that the **Reagent Fraction in Stream** text box is only available when using the non-premixed combustion model with a liquid-phase reagent injection.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Urea Decomposition

allows you to specify the decomposition model to use when the selected **Reagent Species** is **co<nh2>2**.

rate-limiting

specifies that the source terms be calculated according to the rates given in Table 14.3: "Two-Step Urea Breakdown Process".

user-specified

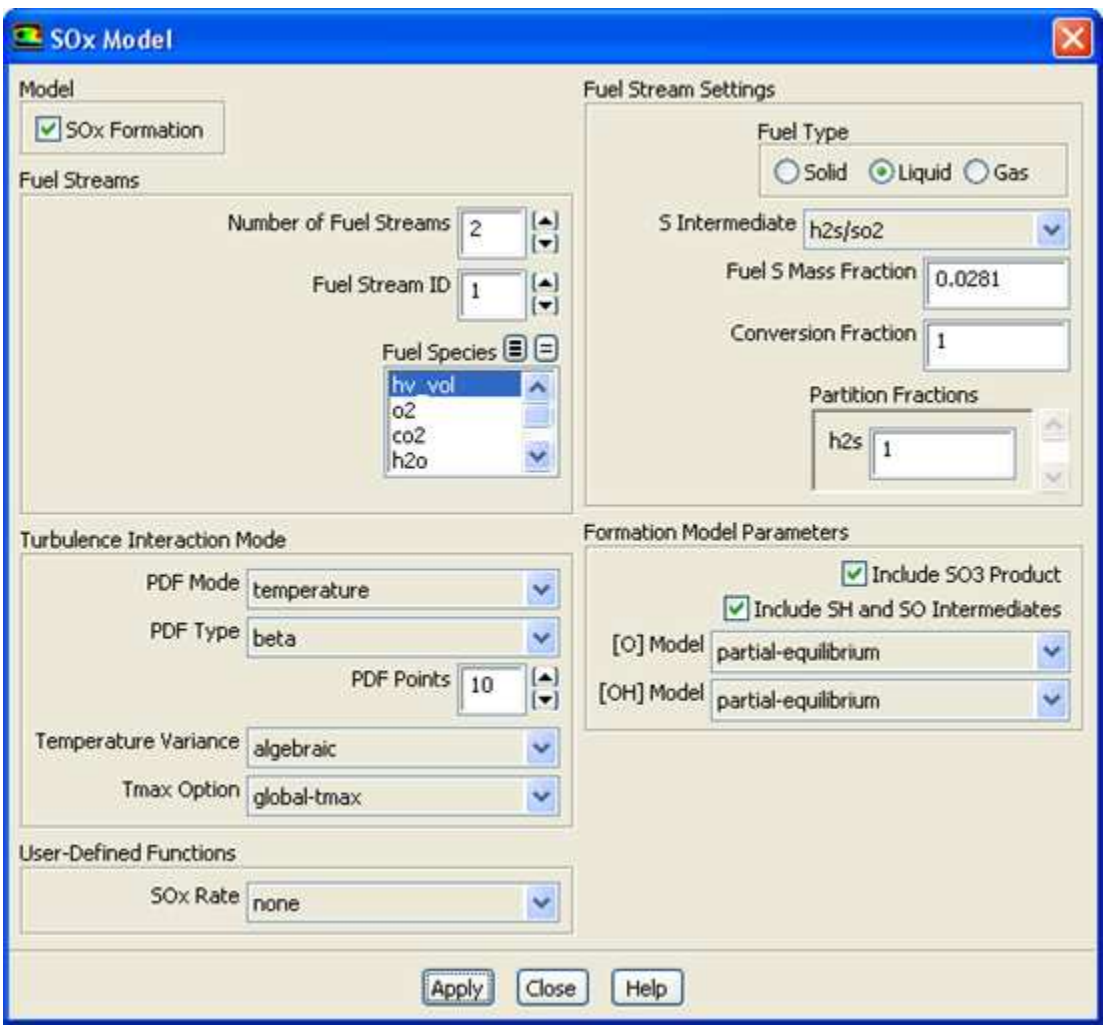
allows you to specify the molar conversion fraction for ammonia, assuming that the rest of the urea is converted to HNCO.

NH3 Conversion

is the mole fraction of NH_3 in the mixture of NH_3 and HNCO instantly created from the reagent injection. The **NH3 Conversion** text box only appears when **user-specified** is selected for **Urea Decomposition**.

35.3.29. SOx Model Dialog Box

The **SOx Model** dialog box allows you to set parameters related to the SOx postprocessor. See *Using the SOx Model* (p. 959) for details about the items below.



Controls

Model

contains the control to enable the model.

SOx Formation

enables the model.

Fuel Streams

allows you to define multiple fuel streams for SOx formation.

Number of Fuel Streams

sets the number of fuel streams. You are allowed up to three fuel streams.

Fuel Stream ID

specifies the fuel stream you are defining in the **Fuel Species** selection list and the **Formation Model Parameters** group box.

Fuel Species

is a list containing all of the defined species. This list is used to specify the species that is the fuel associated with a particular **Fuel Stream ID**, for any combustion model other than non-premixed combustion. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10.

PDF Stream ID

specifies the PDF stream species associated with a particular **Fuel Stream ID**, when calculating SOx formation in conjunction with the non-premixed combustion model.

primary

indicates the primary fuel stream species, as defined in the PDF table.

secondary

indicates the secondary fuel stream species, as defined in the PDF table.

Turbulence Interaction Mode

contains parameters related to the effect of turbulent fluctuations on the SOx formation. See SOx Formation in Turbulent Flows in the Theory Guide for details.

PDF Mode

is a drop-down list containing options that take into account turbulent fluctuations when you compute the specified SO_2 formation. See *Setting Turbulence Parameters* (p. 955) for details.

none

specifies the use of laminar SOx rate calculations, so that the effects of turbulence are ignored.

temperature

includes fluctuations of temperature.

temperature/species

includes fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction

includes fluctuations of the mixture fraction(s). This is available for non-premixed combustion calculations only.

PDF Type

allows you to specify the shape of the PDF.

beta

models the PDF using Equation 14–108 in the Theory Guide.

gaussian

models the PDF using Equation 14–111 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF Points

controls the number of points at which the beta function in Equation 14–105 or Equation 14–106 in the Theory Guide will be integrated. The default value of 10, which indicates that the beta function will be integrated at 10 points on a histogram basis, will yield an accurate solution with reasonable computation time. Increasing this value may improve accuracy, but will also increase the computation time. This text box is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance.

algebraic

is an approximate form of the transport equation (see Equation 14–114 in the Theory Guide).

transported

solves Equation 14–113 in the Theory Guide.

Tmax Option

provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax

sets the limit as the maximum temperature in the flow field.

local-tmax-factor

yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax

sets the limit for each cell to be the value entered in **Tmax**.

user-defined

allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a UDF and selected it in the **SOx Rate** drop-down list in the **User-Defined Functions** group box.

Species

is a drop-down list which appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the SOx rate calculations.

User-Defined Functions

allows you to use a user-defined function (UDF) to contribute to the rate of SOx production. See the separate UDF Manual for details. Note that you may also use a UDF to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

SOx Rate

allows you to select a compiled UDF.

UDF Rate

provides options for the treatment of the SOx production specified by the UDF.

Replace FLUENT Rate

replaces ANSYS FLUENT's SOx rate calculations with the custom SOx rate produced by your UDF.

Add to FLUENT Rate

adds the custom SOx rate produced by your UDF to ANSYS FLUENT's SOx rate calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fuel Stream Settings

contains the parameters associated with a particular fuel stream of the SOx model, as specified in the **Fuel Stream ID** text box in the **Fuel Streams** group box.

Fuel Type

enables selection of the fuel.

Solid

enables the calculation of solid fuel SOx.

Liquid

enables the calculation of liquid fuel SOx.

Gas

enables the calculation of gas fuel SOx.

S Intermediate

drop-down list enables you to select intermediate species (**h2s**, **so2**, or **h2s/so2**).

Volatile S Mass Fraction

specifies the mass fraction of sulfur in the volatiles. This parameter appears only for **Solid** fuel streams.

Fuel S Mass Fraction

field sets the value for correct mass fraction of sulfur in the fuel (kg sulfur per kg fuel). This parameter appears only for **Liquid** and **Gas** fuel streams.

Conversion Fraction

specifies the overall mass fraction of the fuel S (for liquid or gas fuels), or the volatile S or char S (for solid fuels), that will be converted to the intermediate species and/or product SO_2. The **S Intermediate Conversion Fraction** has a default value of 1.

Partition Fractions

specifies the mass fraction of the fuel S (for liquid or gas fuels), or the volatile S or char S (for solid fuels) that will become **h2s**. The remainder will become SO_2. This parameter only appears when you select **h2s/so2** for the **S Intermediate** or **Char S Conversion** drop-down lists.

Char S Conversion

drop-down list selects the char S conversion path as **so2**, **h2s**, or **so2/h2s**. This parameter appears only for **Solid** fuel streams.

Char S Mass Fraction

specifies the mass fraction of sulfur in the char. This parameter appears only for **Solid** fuel streams.

Formation Model Parameters

allows you to include SOx products and intermediates.

Include SO3 Product

includes **SO3** as a product in all of the fuel streams, as described in Reaction Mechanisms for Sulfur Oxidation in the Theory Guide.

Include SH and SO Intermediaries

includes **SH** and **SO** as intermediates in all of the fuel streams, as described in Reaction Mechanisms for Sulfur Oxidation in the Theory Guide.

[O] Model

drop-down list specifies the method by which **O** will be calculated in all of the fuel streams, i.e., **equilibrium**, **partial-equilibrium**, or **instantaneous** in the **[O] Model**.

[OH] Model

drop-down list specifies the method by which **OH** will be calculated in all of the fuel streams, i.e., **equilibrium**, **partial-equilibrium**, or **instantaneous** in the **[OH] Model**.

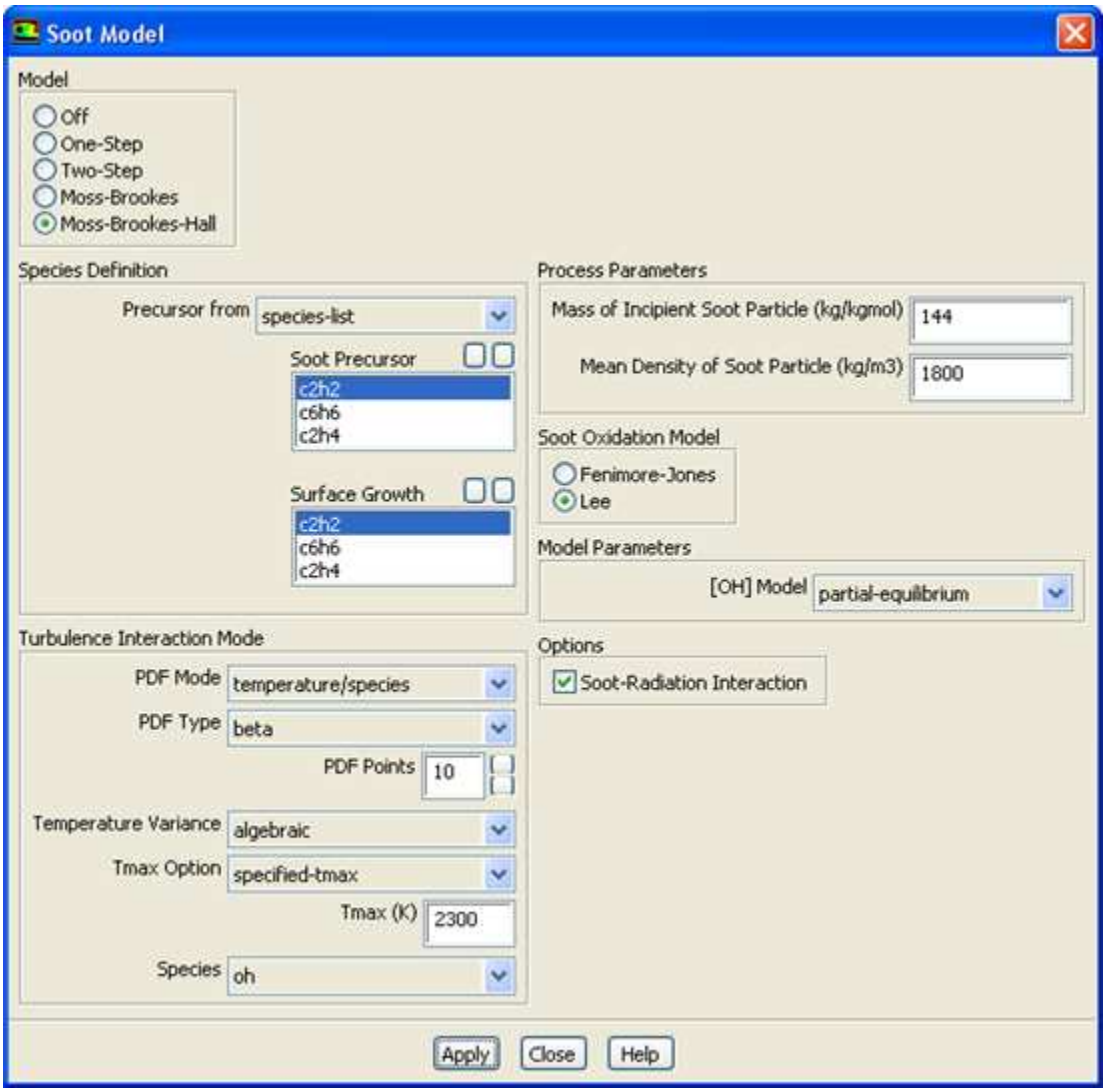
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

35.3.30. Soot Model Dialog Box

The **Soot Model** dialog box allows you to set parameters related to the soot model. See *Using the Soot Models* (p. 973) for details about the items below.



Controls

Model

specifies which model should be used for computing soot formation.

Off

disables the calculation of soot formation.

One-Step

enables the one-step soot model described in The One-Step Soot Formation Model in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Two-Step
enables the two-step soot model described in The Two-Step Soot Formation Model.

Moss-Brookes
enables the Moss-Brookes soot model described in The Moss-Brookes Model in the Theory Guide.

Moss-Brookes-Hall
enables the Moss-Brookes-Hall soot model described in The Moss-Brookes-Hall Model in the Theory Guide. This option is only available when C_2H_2, C_6H_6, C_6H_5, and H_2 are present in the gas phase species list.

Species Definition
contains inputs for specifying the chemical species for your model.

Fuel
is a drop-down list containing all of the defined species. Here you will select the species that is the fuel for the **One-Step** and **Two-Step** models, as well as the **Moss-Brookes** model when a precursor species is not identified in the defined species list.

Oxidant
is a drop-down list containing all of the defined species. Here you will select the species that is the oxidizer for the **One-Step** and **Two-Step** models.

Precursor from
allows you to select from a list of species or enter the correlation values of species. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Soot Precursor
is a selection list containing all of the possible precursor species found via a query of the defined species list. By default, ANSYS FLUENT only considers `c2h2`, `c6h6`, and `c2h4` as possible precursor species. For information about including other species in the possible precursor species search, contact your ANSYS FLUENT support engineer. From this list you will select the species that are the soot precursor species for the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Surface Growth
is a selection list containing all of the possible surface growth species, as explained previously for the **Soot Precursor** selection list. Here you will select the species that are the surface growth species for the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Fuel Carbon Number
is the number of carbon atoms in the species selected in the **Fuel** drop-down list. This text box appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Fuel Hydrogen Number
is the number of hydrogen atoms in the species selected in the **Fuel** drop-down list. This text box appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Molecular Weight of Precursor
is the molecular weight of the precursor species. This text box appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list. The default value is the weight of acetylene.

Precursor Correlation
is a drop-down list you can use to define a laminar diffusion profile which relates mixture fraction to precursor mass fraction. This text box appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

piecewise-polynomial
specifies that the precursor mass fraction is a piecewise-polynomial function of mixture fraction. The default values used by ANSYS FLUENT correspond to a methane diffusion flame simulation, in which both the air and fuel initial temperatures are set to 290 K, and acetylene is assumed as the soot precursor. These values can be revised via the **Edit...** button.

constant
specifies that the precursor mass fraction is a constant function of mixture fraction, the value of which is specified in the text box below the **Precursor Correlation** drop-down list.

Edit...
opens the *Piecewise-Polynomial Profile Dialog Box* (p. 1752)when **piecewise-polynomial** is selected from the **Precursor Correlation** drop-down list, thus allowing you to revise the default values.

Turbulence Interaction Mode
contains inputs that specify how turbulent fluctuations are accounted for in the soot formation calculations for the **Moss-Brookes** and **Moss-Brookes-Hall** models. For further details on these inputs, see *Setting Up the Moss-Brookes Model and the Hall Extension* (p. 978).

PDF Mode
is a drop-down list that contains the options for addressing turbulent fluctuations in the soot rate calculations. Note that **mixture fraction** is the most accurate option, and should be used if it is available.

none
specifies the use of laminar soot rate calculations, so that the effects of turbulence are ignored.

temperature
specifies that the soot rate calculations include the effect of temperature fluctuations.

temperature/species
specifies that the soot rate calculations include the effect of fluctuations of temperature, as well as fluctuations of the mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction
is the most accurate option, specifying that the soot rate calculations include the effect of fluctuations of mixture fraction(s). Note that this option is not available if you are using the eddy-dissipation model.

PDF Type
allows you to specify the shape of the PDF.

beta
models the PDF using Equation 14–108 in the Theory Guide.

gaussian
models the PDF using Equation 14–111 in the Theory Guide.

PDF Points
controls the number of points at which the beta function will be integrated on a histogram basis. Increasing this number may improve accuracy, but will also increase compute time. This text box is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance
allows you to specify the form of transport equation that is solved to calculate the temperature variance.

algebraic
is an approximate form of the transport equation (see Equation 14–114 in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

transported
solves Equation 14–113 in the Theory Guide.

Tmax Option
provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax
sets the limit as the maximum temperature in the flow field.

local-tmax-factor
yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax
sets the limit for each cell to be the value entered in **Tmax**.

user-defined
allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a UDF.

Species
is a drop-down list which appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the soot rate calculations.

Process Parameters
contains parameters that control the combustion process modeling.

Mean Diameter of Soot Particle
is the assumed average diameter of the soot particles in the combustion system, used to compute the soot particle mass m_p in Equation 14–134 in the Theory Guide for the **Two-Step** model.

Mean Density of Soot Particle
is the assumed average density of the soot particles in the combustion system. For the **Two-Step** model, it is used to compute the soot particle mass m_p in Equation 14–134 in the Theory Guide. For the **Moss-Brookes** and **Moss-Brookes-Hall** models, it is ρ_{soot} in Equation 14–141 and ρ in Equation 14–144 in the Theory Guide. The default value supplied by ANSYS FLUENT is 1800 kg/m^3 (as was used in the work of Brookes and Moss [*12*] (p. 2209)).

Stoichiometry for Soot Combustion
is the mass stoichiometry v_{soot} in Equation 14–131 in the Theory Guide, which computes the soot combustion rate in the **One-Step** and **Two-Step** models. The default value supplied by ANSYS FLUENT (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion
is the mass stoichiometry v_{fuel} in Equation 14–131 in the Theory Guide, which computes the soot combustion rate in the **One-Step** and **Two-Step** models. The default value supplied by ANSYS FLUENT (3.6363) is for combustion of propane (C_3H_8) by oxygen (O_2).

Mass of Incipient Soot Particle
is M_P in Equation 14–142 and Equation 14–144, which is used in the **Moss-Brookes** and **Moss-Brookes-Hall** model computations. The default value supplied by ANSYS FLUENT (144 kg/mol) is the mass of 12 carbon atoms. Note that for the original implementation of the Hall extension, the model assumed this mass to be 100 carbon atoms.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Soot Oxidation Model

contains model options that determine the form of the soot oxidation term in the calculations of the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Fenimore-Jones

takes into account the soot oxidation due to the hydroxyl radical.

Lee

takes into account the soot oxidation due to the hydroxyl radical and molecular oxygen.

Model Parameters

contains parameters that control the soot formation model.

Soot Formation Constant

is the parameter C_s in Equation 14–128 in the Theory Guide. This item appears only for the **One-Step** soot model.

Equivalence Ratio Exponent

is the exponent r in Equation 14–128 in the Theory Guide. This item appears only for the **One-Step** soot model.

Equivalence Ratio Minimum

and **Equivalence Ratio Maximum** are the minimum and maximum values of the fuel equivalence ratio ϕ in Equation 14–128 in the Theory Guide. Equation 14–128 will be solved only if **Equivalence Ratio Minimum**$< \phi <$ **Equivalence Ratio Maximum**; if ϕ is outside of this range, there is no soot formation. This item appears only for the **One-Step** soot model.

Activation Temperature of Soot Formation Rate

is the term E/R in Equation 14–128 in the Theory Guide. This item appears only for the **One-Step** soot model.

Magnussen Constant for Soot Combustion

is the constant A used in the rate expressions governing the soot combustion rate (Equation 14–130 and Equation 14–131) in the Theory Guide. This item appears only for the **One-Step** and the **Two-Step** soot models. For the **Two-Step** model, this input will be called **Magnussen Constant for Soot and Nuclei Combustion**.

Limiting Nuclei Formation Rate

is the limiting value of the kinetic nuclei formation rate η_0 in Equation 14–137 in the Theory Guide.

Below this limiting value, the branching and termination term, $(f-g)$ in Equation 14–136 in the Theory Guide, is not included. This item appears only for the **Two-Step** soot model.

Nuclei Branching-Termination Coefficient

is the term $(f-g)$ in Equation 14–136 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Nuclei Coefficient of Linear Termination on Soot

is the term g_0 in Equation 14–136 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Pre-Exponential Constant of Nuclei Formation

is the pre-exponential term a_0 in the kinetic nuclei formation term, Equation 14–137 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Activation Temperature of Nuclei Formation Rate

is the term E/R in the kinetic nuclei formation term, Equation 14–137 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alpha for Soot Formation Rate
is α, the constant in the soot formation rate equation, Equation 14–134 **Two-Step** soot model.

Beta for Soot Formation Rate
is β, the constant in the soot formation rate equation, Equation 14–134 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Magnussen Constant for Soot and Nuclei Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14–130 and Equation 14–131 in the Theory Guide). This item appears only for the **One-Step** and the **Two-Step** soot models.

[OH] Model
is a drop-down list that allows you to specify the method by which the OH radical concentration is calculated, i.e., **instantaneous** or **partial-equilibrium**. This list appears only for the **Moss-Brookes** and the **Moss-Brookes-Hall** soot models.

[O] Model
is a drop-down list that specifies the method by which the O radical concentration is calculated, i.e., **equilibrium**, **partial-equilibrium**, or **instantaneous**. This list appears only for the **Moss-Brookes** and the **Moss-Brookes-Hall** soot models, when you have selected **partial-equilibrium** for the **[OH] Model**.

Important

To use the concentration of OH or O predicted by the combustion model, select **instantaneous** for **[OH] Model**or **[O] Model**.

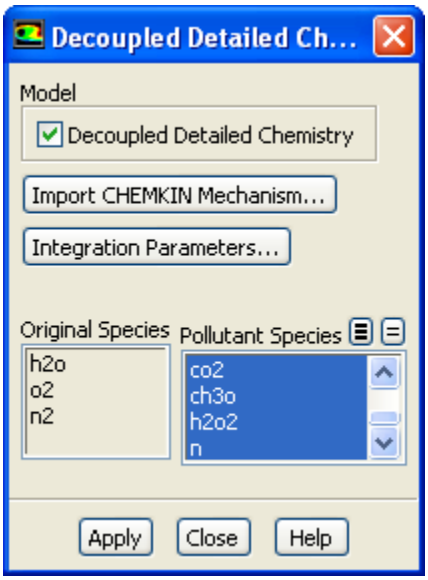
Options
contains an option for modeling the effect of soot on a variable radiation absorption coefficient. This group box will appear only when one of the radiation models in the *Radiation Model Dialog Box* (p. 1657) is active.

Soot-Radiation Interaction
enables the soot-radiation interaction model described in The Effect of Soot on the Absorption Coefficient in the Theory Guide.

35.3.31. Decoupled Detailed Chemistry Dialog Box

The **Decoupled Detailed Chemistry** dialog box allows you to postprocess slowly-forming, trace pollutant species on a steady-state flow field using detailed chemical kinetic mechanisms.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model
includes the option to enable **Decoupled Detailed Chemistry**.

Import CHEMKIN Mechanism...
opens the *CHEMKIN Mechanism Import Dialog Box* (p. 2055).

Integration Parameters...
opens the *Integration Parameters Dialog Box* (p. 1694).

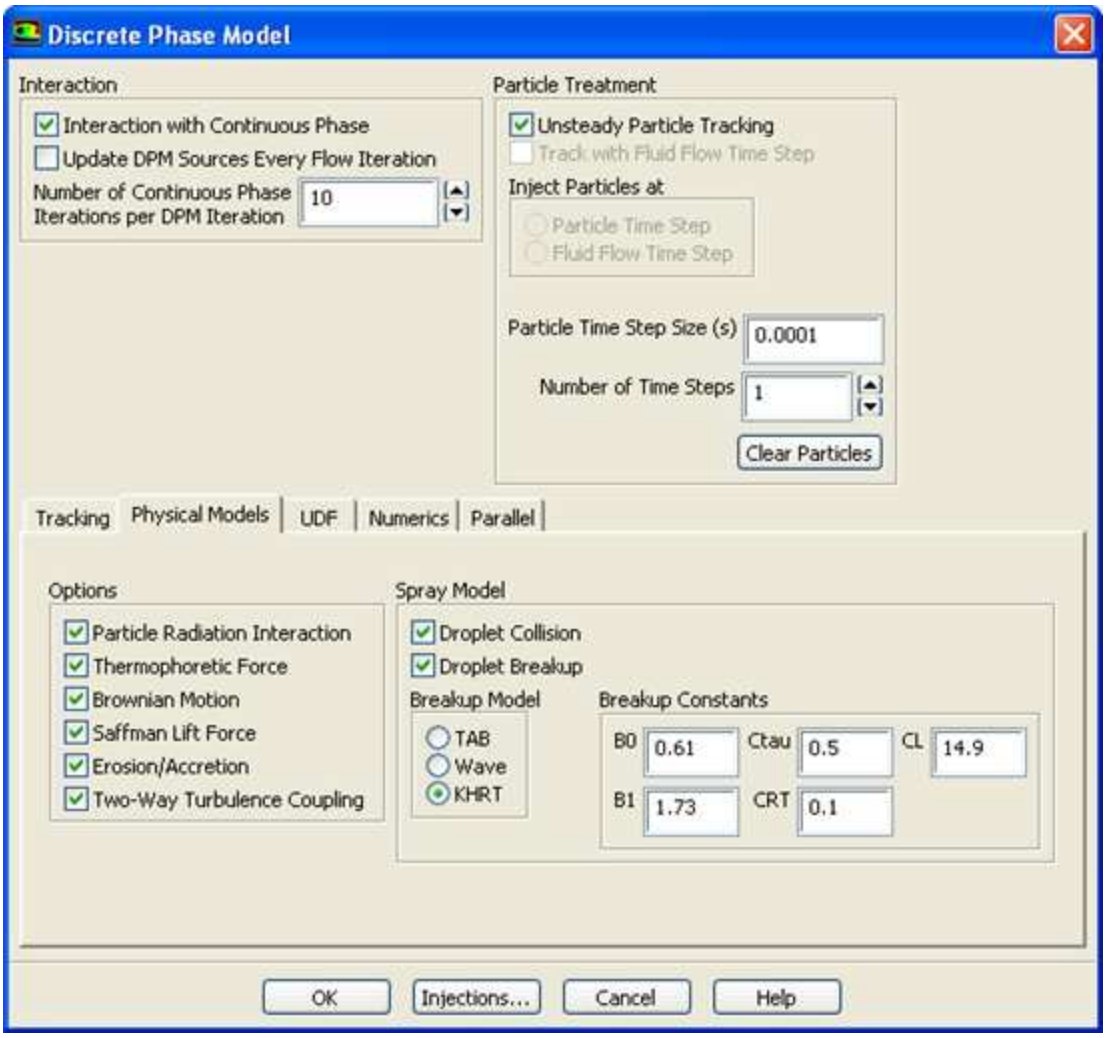
Original Species
are the species that are in the original case setup.

Pollutant Species
are typically slowly forming (far from chemical equilibrium), and occur at miniscule mass fractions.

35.3.32. Discrete Phase Model Dialog Box

The **Discrete Phase Model** dialog box allows you to set parameters related to the calculation of a discrete phase of particles. See *Modeling Discrete Phase* (p. 1007) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Interaction

contains parameters used for performing coupled calculations of the continuous and discrete phase flow. See *Procedures for a Coupled Two-Phase Flow* (p. 1065) for details.

Interaction with Continuous Phase

enables a coupled calculation of the discrete phase and the continuous phase.

Update DPM Sources Every Flow Iteration

enables calculation of particle source terms at every DPM Iteration. It is recommended for unsteady simulations.

Number of Continuous Phase Iterations per DPM Iteration

allows you to control the frequency at which the particles are tracked and the DPM sources are updated.

Particle Treatment

contains options for choosing to treat the particles in an unsteady or a steady fashion.

Unsteady Particle Tracking

enables unsteady tracking of particles.

Track with Fluid Flow Time Step

enables the use of fluid flow time steps to inject the particles.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inject Particles at
contains parameters to decide when to inject the particles for a new time step.

Particle Time Step
enables injection of particles for every particle time step.

Fluid Flow Time Step
enables injection of particles for every fluid flow time step. In any case, the particles will always be tracked in such a way that they coincide with the flow time of the continuous flow solver.

Particle Time Step Size
specifies particle time step size for the calculation.

Number of Time Steps
allows you to specify the number of time steps for the calculation.

Clear Particles
clears the particles that are currently in the domain.

Tracking
contains two parameters to control the time integration of the particle trajectory equations.

Tracking Parameters
contains parameters that control the tracking of particle trajectories. One simple rule of thumb to follow when setting the two parameters below is that if you want the particles to advance through a domain of length D, the **Length Scale** times the number of **Max. Number of Steps** should be approximately equal to D. See Integration of Particle Equation of Motion in the Theory Guide for details about the items below.

Max. Number of Steps
is the maximum number of time steps used to compute a single particle trajectory via integration of Equation 16–1 in the Theory Guide and Equation 16–50.

Specify Length Scale
when enabled allows you to specify the length scale.

Length Scale
controls the integration time step size used to integrate the equations of motion for the particle. It appears only when the **Specify Length Scale** option is enabled.

Step Length Factor
specifies the value of λ in *Equation 25–2* (p. 1014).

Drag Parameters
allows the setting of the drag law used in calculating the force balance on the particles. See Particle Force Balance in the Theory Guide for details on the items below.

Drag Law
is a drop-down list containing three choices:

spherical
assumes that the particles are smooth spheres.

nonspherical
assumes that the particles are not spheres, but are all identically shaped. The shape is specified by the **Shape Factor**.

Stokes-Cunningham
is for use with sub-micron particles. A **Cunningham Correction** is added to Stokes' drag law to determine the drag.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

high-Mach-number
is similar to the spherical law with corrections to account for a particle Mach number greater than 0.4 or a particle Reynolds number greater than 20.

dynamic-drag
accounts for the effects of droplet distortion. This drag law is available only when one of the droplet breakup models is used in conjunction with unsteady tracking. See Dynamic Drag Model Theory in the Theory Guide for details.

Shape Factor
specifies the shape of the particles when **nonspherical** is selected as the **Drag Law** (ϕ in Equation 16–66 in the Theory Guide). It is the ratio of the surface area of a sphere having the same volume as the particle to the actual surface area of the particle. The shape factor value cannot be greater than 1.

Cunningham Correction
(C_c in Equation 16–68 in the Theory Guide) is used with Stokes' drag law to determine the force acting on the particles when the particles are sub-micron size. It appears when **Stokes-Cunningham** is selected as the **Drag Law**.

Physical Models
contains optional discrete phase models and their relevant parameters.

Options
contains additional models that can be included in the calculation. See *Physical Models for the Discrete Phase Model* (p. 1015) for more information.

Particle Radiation Interaction
includes the effect of radiation heat transfer to the particles (Equation 5–34 in the Theory Guide). You will also need to define additional properties for the particle materials (emissivity and scattering factor), as described in *Description of the Properties* (p. 1059).

This item appears only if the P-1 or discrete ordinates model is selected in the *Radiation Model Dialog Box* (p. 1657).

Thermophoretic Force
enables the inclusion of a thermophoretic force on the particles as an additional force term. See Thermophoretic Force in the Theory Guide for details.

Brownian Motion
enables the incorporation of the effects of Brownian motion. See Brownian Force in the Theory Guide for details.

Saffman Lift Force
enables the inclusion of Saffman's lift force (lift due to shear) as an additional force term. See Saffman's Lift Force in the Theory Guide for details.

Erosion/Accretion
enables the monitoring of erosion/accretion rates at wall boundaries. See *Monitoring Erosion/Accretion of Particles at Walls* (p. 1017) for details. This item appears only if **Interaction with Continuous Phase** is enabled.

Two-Way Turbulence Coupling
enables the effect of change in turbulent quantities due to particle damping and turbulence eddies.

Spray Model
contains parameters that control droplet breakup and collision. (This section of the dialog box appears only if **Unsteady Tracking** is enabled.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Droplet Collision
includes the effect of droplet collisions. See Droplet Collision and Coalescence Model Theory in the Theory Guide for details.

Droplet Breakup
includes the effects of droplet breakup.

Breakup Model
contains parameters that control droplet breakup. (This item appears only if **Droplet Breakup** is enabled.)

TAB
enables the Taylor Analogy Breakup (TAB) model, which is applicable to many engineering sprays. This method is based upon Taylor's analogy between an oscillating and distorting droplet and a spring mass system. See Taylor Analogy Breakup (TAB) Model in the Theory Guide for details.

Wave
enables the Wave breakup model, which considers the breakup of the injected liquid to be induced by the relative velocity between the gas and liquid phases. See Wave Breakup Model for details.

KHRT
enables the Kelvin-Helmholtz Rayleigh-Taylor breakup model, which considers the two competing effects of aerodynamic breakup and instabilities due to droplet acceleration. See KHRT Breakup Model for details.

Breakup Constants
contains model constants used in the equations for spray breakup. (This item appears only if **Droplet Breakup** is enabled.)

y0
(only for the TAB model) is the constant y_0 in Equation 16–258 in the Theory Guide.

Breakup Parcels
is the number of child parcels the droplet is split into, as described in Velocity of Child Droplets in the Theory Guide.

B0
(only for the Wave and KHRT models) is the constant B_0 in Equation 16–284 in the Theory Guide.

B1
(only for the Wave and KHRT models) is the constant B_1 in Equation 16–286 in the Theory Guide.

Ctau
(only for the KHRT model) is the constant C_τ in Equation 16–291 in the Theory Guide.

CRT
(only for the KHRT model) is the constant C_{RT} in Equation 16–292 in the Theory Guide.

CL
(only for the KHRT model) is the constant C_L in Equation 16–287 in the Theory Guide.

UDF
contains parameters that can be used to customize the discrete phase model using UDF. See the separate UDF Manual for details about user-defined functions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Functions
lists will show available user-defined functions that can be selected to customize the discrete phase model.

Body Force
contains a drop-down list of user-defined functions available for including additional body forces.

Erosion/Accretion
contains a drop-down list of user-defined functions available for incorporating non-standard erosion rate definitions. This itemwill appear only when the **Erosion/Accretion** option has been enabled.

Scalar Update
contains a drop-down list of user-defined functions available for calculating or integrating scalar values along the particle trajectory.

Source
contains a drop-down list of user-defined functions available for modifying interphase exchange terms.

Spray Collide Function
contains a drop-down list of user-defined functions available for modifying spray collide function.

DPM Timestep
contains a drop-down list of user-defined functions available for modifying DPM time step.

User Variables
lists the user input variables.

Number of Scalars
sets the number of scalar values used in the calculations with user-defined functions.

Numerics
tab gives you control over the numerical schemes for particle tracking as well as solutions of heat and mass equations. See *Numerics of the Discrete Phase Model* (p. 1019) for details.

Options
contains parameters of solutions of heat and mass equations.

Accuracy Control
enables the solution of equations of motion within a specified tolerance.

Tolerance
is the maximum relative error which has to be achieved by the tracking procedure.

Max. Refinements
is the maximum number of step size refinements in one single integration step. If this number is exceeded the integration will be conducted with the last refined integration step size.

Coupled Heat-Mass Solution
enables the solution of the particle heat and mass equations using a stiff, coupled ODE solver with error tolerance control. See *Including Coupled Heat-Mass Solution Effects on the Particles* (p. 1021) for details.

Track in Absolute Frame
enables tracking the particles in the absolute reference frame.

Tracking Scheme Selection
contains parameters for selection of numerical schemes.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automated
enables a mechanism to switch in an automated fashion between numerically stable lower order schemes and higher order schemes, which are stable only in a limited range.

High Order Scheme
can be chosen from the group consisting of **trapezoidal** and **runge-kutta** scheme.

Low Order Scheme
consists of **implicit** and the exponential **analytic** integration scheme.

Tracking Scheme
allows to choose any of the tracking schemes. You also can combine each of the tracking schemes with Accuracy Control. It is selectable only if **Automated** is switched off.

Parallel
contains parameters that control the compute nodes for performing discrete phase calculations in parallel. See *Parallel Processing for the Discrete Phase Model* (p. 1087) for details.

Methods
allows you to select the mode for parallel processing. This group box is editable only for the parallel version of ANSYS FLUENT; for the serial version, the **Shared Memory** option (see the description that follows) is selected by default.

Message Passing
enables cluster computing and also works on shared memory machines. With this option, the compute node processes themselves perform the particle work on their local partitions and particle migration to other compute nodes is implemented using message passing primitives. No special requirements are placed on the host machine. Note that this model is not available if the **Cloud Model** option is enabled in the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box. The **Message Passing** option is irrelevant for the serial ANSYS FLUENT code while the **Shared Memory** option remains valid.

Shared Memory
allows you to specify parameters for performing the calculations on shared-memory multiprocessor machines. Note that this option is not available on Windows 2000 or the `ntx86` platform (32–bit Windows).

Hybrid
allows you to combine **Message Passing** (as previously described) and **OpenMP** for a dynamic load balancing without migration of cells. This option enables multicore cluster computing, and also works on shared-memory machines. When using the **Hybrid** option, you can control the maximum number of threads on each machine via the **Thread Control** dialog box (see *Controlling the Threads* (p. 1625) for details). Note that this option is not available for the `ntx86` platform (32–bit Windows), nor is it available if the **Cloud Model** option is enabled in the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box.

Shared Memory Options
contains settings for the **Shared Memory** option. This group box is only available if **Shared Memory** is selected from the **Methods** list.

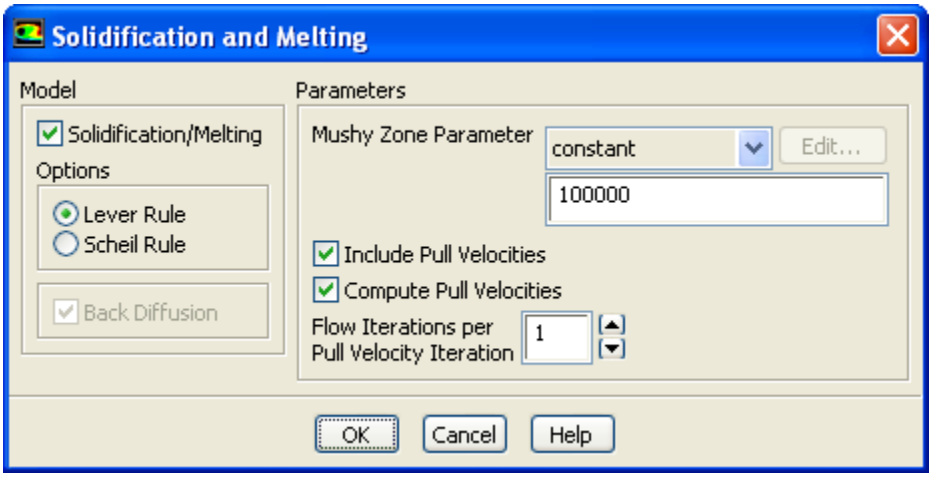
Workpile Algorithm
specifies that the discrete phase calculations are to be performed on a shared-memory multiprocessor machine.

Number of Threads
specifies the number of threads to be used in performing the particle calculations. By default, this parameter is equal to the number of compute nodes you specified for the parallel solver. (This item appears only if **Workpile Algorithm** is enabled.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3.33. Solidification and Melting Dialog Box

The **Solidification and Melting** dialog box allows you to set parameters related to the solidification/melting model. See *Modeling Solidification and Melting* (p. 1201) for details about the items below.



Controls

Model

contains the option for turning on the model.

Solidification/Melting

enables/disables the modeling of solidification and/or melting.

Options

gives you the option of selecting one of two model rules.

Lever Rule

allows you to use the Lever rule (Equation 18–14 in the Theory Guide).

Scheil Rule

allows you to use the Scheil rule (Equation 18–18 in the Theory Guide). If you select **Scheil Rule**, then you can enable **Back Diffusion**. Enter either a **constant** or a user defined function to specify the value of the **Back Diffusion Parameter**, used in Equation 18–19 to Equation 18–21 in the Theory Guide. Please refer to DEFINE_SOLIDIFICATION_PARAMS of the UDF Manual for detailed information about the user defined function.

Parameters

contains parameters related to the solidification/melting model.

Mushy Zone Constant

sets the value of A_{mush} in Equation 18–6 in the Theory Guide.

Include Pull Velocities

includes pull velocities in the model, as described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide.

Compute Pull Velocities

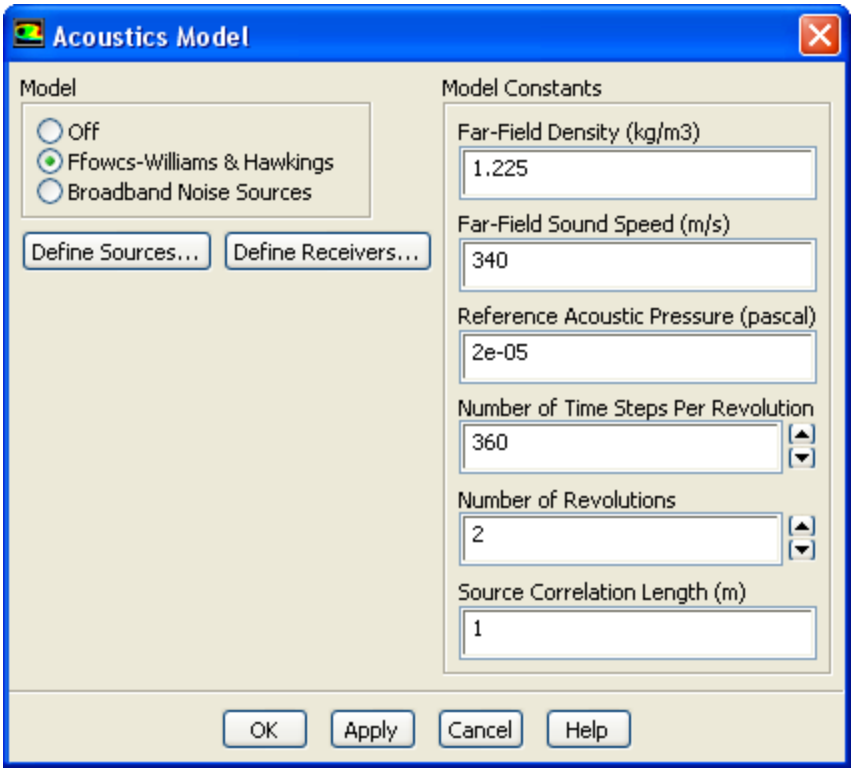
enables the calculation of pull velocities based on the specified velocity boundary conditions, as described in Pull Velocity for Continuous Casting in the Theory Guide. This option appears when **Include Pull Velocities** is turned on.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow Iterations per Pull Velocity Iteration
specifies the number of times the pull velocity equations will be solved after each iteration of the solver. This option appears when **Compute Pull Velocities** is turned on.

35.3.34. Acoustics Model Dialog Box

The **Acoustics Models** dialog box allows you to set parameters related to the acoustics model. See *Using the Ffowcs Williams and Hawkings Acoustics Model* (p. 989) for details about the items below.



Controls

Model
contains the option for turning on the acoustics model.

Off
disables the acoustics model.

Ffowcs-Williams & Hawkings
enables the Ffowcs Williams and Hawkings (FW-H) acoustics model. See *Postprocessing the FW-H Acoustics Model Data* (p. 1000) for details.

Broadband Noise Sources
enables the broadband noise acoustics model. See *Using the Broadband Noise Source Models* (p. 1003) for details.

Model Constants
contains parameters related to the acoustics model.

Far-Field Density
sets the value of the far-field fluid density (ρ_0 in Equation 15–1 in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Far-Field Sound Speed
sets the speed of the sound at the far-field (a_0 in Equation 15–5 and Equation 15–6 in the Theory Guide).

Reference Acoustic Pressure
is used to calculate the sound pressure level in dB.

Reference Acoustic Power
is used to non-dimensionalize the acoustic power, giving the sound pressure level in dB (see *Fast Fourier Transform (FFT) Postprocessing* (p. 1500)). This parameter is equal to 4×10^{-10} W for airborne sound and 10^{-12} W for underwater sound.

Reference Acoustic Intensity
is defined as $p_{ref}^2/\left(\rho_0 a_0\right)$, although this quantity is not currently used by ANSYS FLUENT.

Source Correlation Length
is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depthwise direction using the identical source data.

Number of Realizations
is the number of times the noise source terms will be computed through the generation of stochastic turbulent velocity field.

Number of Fourier Modes
is the number of terms in the Fourier summation from which the turbulent velocity field and its derivatives are computed.

Number of Time Steps Per Revolution
is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.

Number of Revolutions
is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of revolutions that will be simulated in the model.

Options
contains the parameters related to **Ffowcs-Williams & Hawkings** model.

Export Acoustic Source Data in ASD Format
enables/disables the saving of source data files in ASD format.

Export Acoustic Source Data in CGNS Format
enables/disables the saving of source data files in CGNS format.

Compute Acoustics Signals Simultaneously
enables "on-the-fly" calculation of sound. When this option is chosen, the ANSYS FLUENT console window will print a message at the end of each time step indicating that the sound pressure signals have been computed (e.g., `Extracting sound signals at x receiver locations...`, where `x` is the number of receivers you specified). Enabling this option instructs ANSYS FLUENT to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Define Sources...
opens the *Acoustic Sources Dialog Box* (p. 1733), which allows you to specify the acoustics sources.

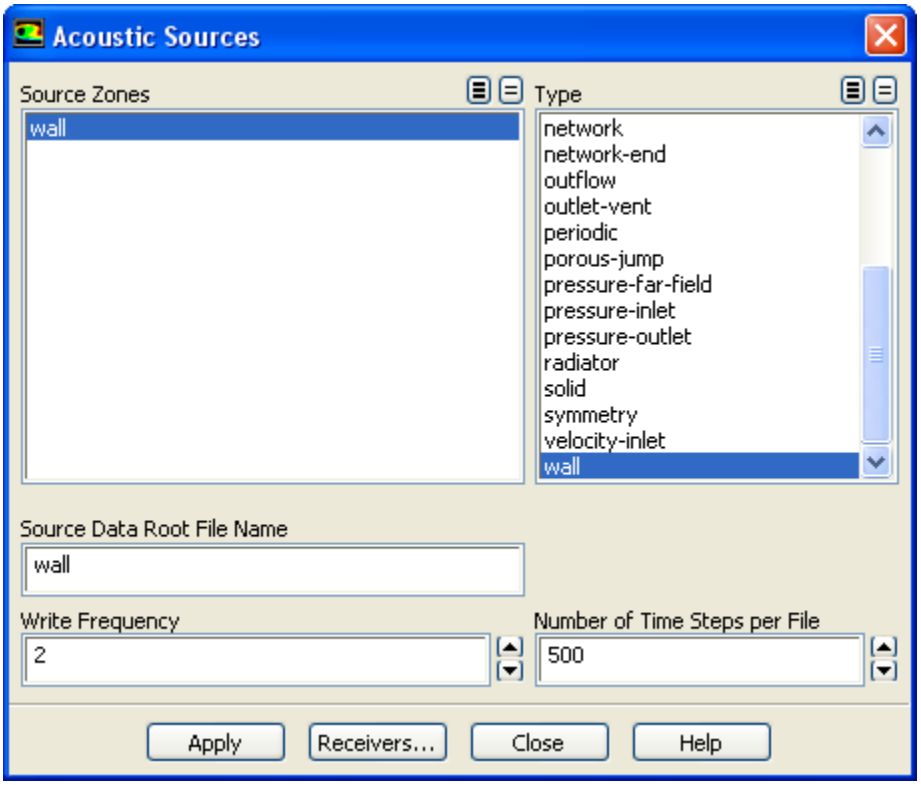
Define Receivers...
opens the *Acoustic Receivers Dialog Box* (p. 1734), which allows you to specify the location of the receivers.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply
saves the acoustics model settings.

35.3.35. Acoustic Sources Dialog Box

The **Acoustic Sources** dialog box allows you to specify the acoustics sources. See *Using the Ffowcs Williams and Hawkings Acoustics Model* (p. 989) for details about the items below.



Controls

Source Zones
contains a list of zones from which you can select one or more emission (source) surfaces.

Type
contains a list of all possible types of zones from which you can select one or more types of the available zones. If you select an available type of zone, the relevant zones will then be selected in the **Source Zones** list. If you specify any valid **interior** zones as source surfaces, the *Interior Cell Zone Selection Dialog Box* (p. 1734) will appear.

Source Data Root File Name
is used to give the names of the source data files (e.g., `acoustic_example_xxxx.asd`, where `xxxx`is the global time-step index of the transient solution) and an index file (e.g., `acoustic_example.index`) that will store the information associated with the source data.

Write Frequency
allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict.

Number of Time Steps per File
allows you to write the source data to multiple files, each containing the source data for a number of time steps specified by you.

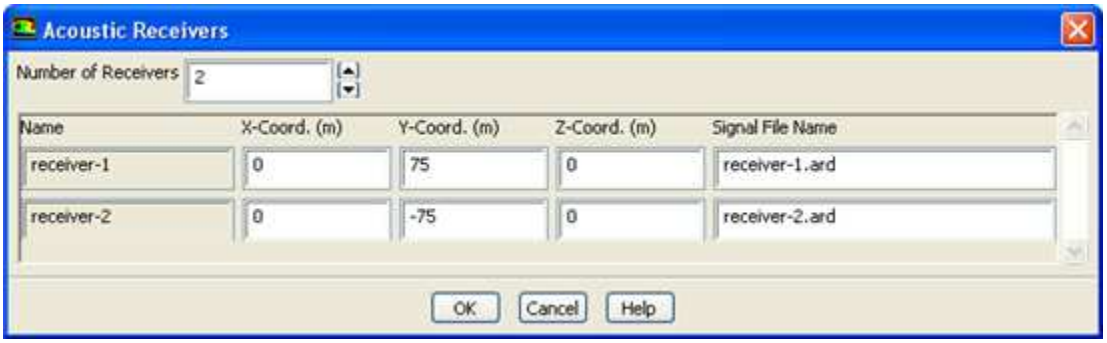
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Receivers...

opens the *Acoustic Receivers Dialog Box* (p. 1734), which allows you to specify the location of the receivers.

35.3.36. Acoustic Receivers Dialog Box

The **Acoustic Receivers** dialog box allows you to specify the location of the acoustic receivers. See *Using the Ffowcs Williams and Hawkings Acoustics Model* (p. 989) for details about the items below.



Controls

Number of Receivers

allows you to specify the total number of receivers for which you want to compute sound.

Name

shows the name of the receiver. If you edit the field, the new name will take effect after you click the **OK** button.

X-Coord., Y-Coord., Z-Coord.

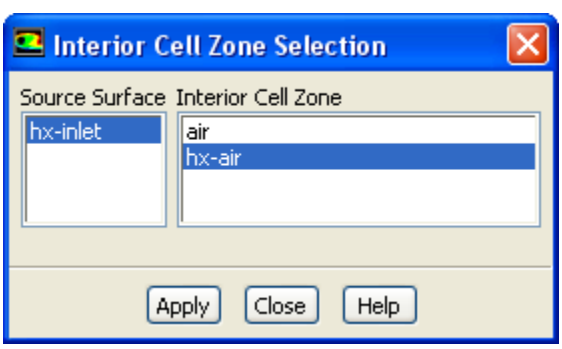
specifies the coordinates for each receiver. Note that because ANSYS FLUENT's acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (i.e., the selected acoustic surface), and can fall outside of the computational domain if needed.

Signal File Name

specifies the name of the file used to store sound pressure signals for the corresponding receivers. By default, the files will be named `receiver-1.dat`, `receiver-2.dat`, etc.

35.3.37. Interior Cell Zone Selection Dialog Box

The **Interior Cell Zone Selection** dialog box allows you to specify the acoustics sources. See *Specifying Source Surfaces* (p. 995) for details about the items below.



Controls

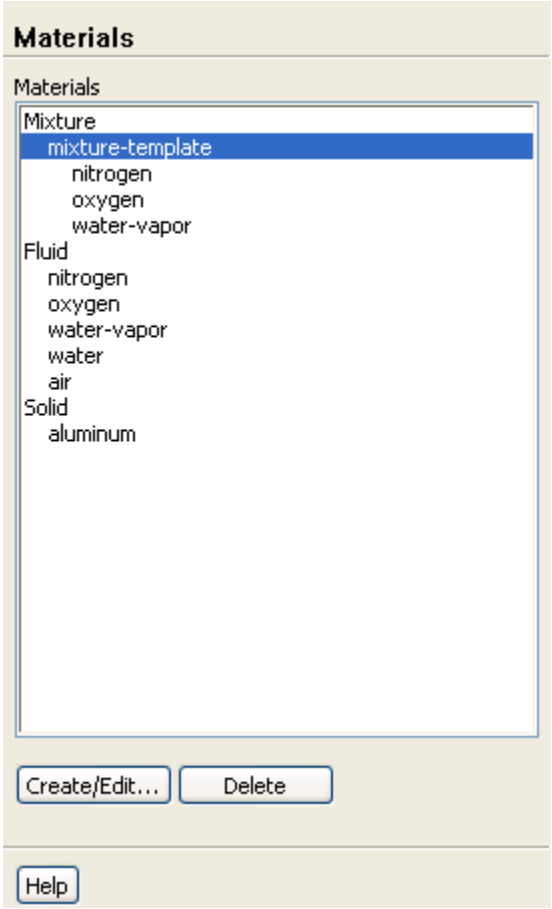
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Source Surface
displays the selected interior surface.

Interior Cell Zone
contains a list of two interior cell zones, from which you will select the zone that is occupied by the quadrupole sources.

35.4. Materials Task Page

The **Materials** task page allows you to set properties for any fluid or solid (or mixture, if applicable) materials in your ANSYS FLUENT simulation.



Controls

Materials
contains a listing of available fluid or solid (or mixture, if applicable) materials.

Create/Edit...
displays the *Create/Edit Materials Dialog Box* (p. 1736) for the selected item in the **Materials** list.

Delete
removes the selected material from the **Materials** list,

For additional information, please see the following sections:

35.4.1. Create/Edit Materials Dialog Box
35.4.2. FLUENT Database Materials Dialog Box
35.4.3. Open Database Dialog Box
35.4.4. User-Defined Database Materials Dialog Box

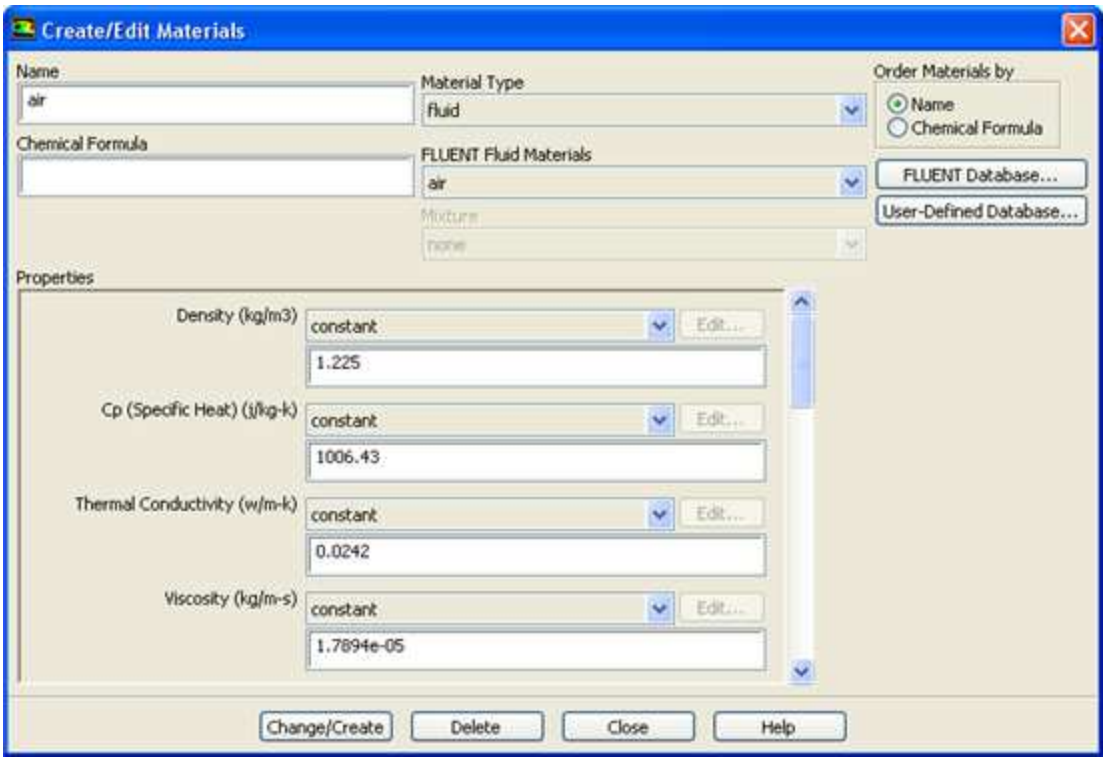
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.5. Copy Case Material Dialog Box
35.4.6. Material Properties Dialog Box
35.4.7. Edit Property Methods Dialog Box
35.4.8. New Material Name Dialog Box
35.4.9. Polynomial Profile Dialog Box
35.4.10. Piecewise-Linear Profile Dialog Box
35.4.11. Piecewise-Polynomial Profile Dialog Box
35.4.12. User-Defined Functions Dialog Box
35.4.13. Sutherland Law Dialog Box
35.4.14. Power Law Dialog Box
35.4.15. Non-Newtonian Power Law Dialog Box
35.4.16. Carreau Model Dialog Box
35.4.17. Cross Model Dialog Box
35.4.18. Herschel-Bulkley Dialog Box
35.4.19. Biaxial Conductivity Dialog Box
35.4.20. Cylindrical Orthotropic Conductivity Dialog Box
35.4.21. Orthotropic Conductivity Dialog Box
35.4.22. Anisotropic Conductivity Dialog Box
35.4.23. Species Dialog Box
35.4.24. Reactions Dialog Box
35.4.25. Third-Body Efficiencies Dialog Box
35.4.26. Pressure-Dependent Reaction Dialog Box
35.4.27. Coverage-Dependent Reaction Dialog Box
35.4.28. Reaction Mechanisms Dialog Box
35.4.29. Site Parameters Dialog Box
35.4.30. Mass Diffusion Coefficients Dialog Box
35.4.31. Thermal Diffusion Coefficients Dialog Box
35.4.32. UDS Diffusion Coefficients Dialog Box
35.4.33. WSGGM User Specified Dialog Box
35.4.34. Gray-Band Absorption Coefficient Dialog Box
35.4.35. Delta-Eddington Scattering Function Dialog Box
35.4.36. Gray-Band Refractive Index Dialog Box
35.4.37. Single Rate Devolatilization Dialog Box
35.4.38. Two Competing Rates Model Dialog Box
35.4.39. CPD Model Dialog Box
35.4.40. Kinetics/Diffusion-Limited Combustion Model Dialog Box
35.4.41. Intrinsic Combustion Model Dialog Box
35.4.42. Edit Material Dialog Box
35.4.43. Fluent Database Materials Dialog Box

35.4.1. Create/Edit Materials Dialog Box

The **Create/Edit Materials** dialog box is used to create and modify materials. Materials can be downloaded from the global database or defined locally. See *Physical Properties* (p. 385) for details about defining material properties. *Using the Materials Task Page* (p. 387) describes how to use the dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

shows the name of the material. If you edit this field, the new name will take effect when you click on **Change/Create**.

Chemical Formula

displays the chemical formula for the material. You should generally not edit this field unless you are creating a material from scratch.

Material Type

is a drop-down list containing all of the available material types. By default, **fluid** and **solid** will be the only choices. If you are modeling species transport/combustion, **mixture** will also be available. For problems in which you have defined discrete-phase injections, **inert-particle**, **droplet-particle**, and/or **combusting-particle** will also appear.

FLUENT Fluid Materials

allows you to choose the fluid material for which you want to modify properties. This option is available when **fluid** is selected in the **Material Type** drop-down list.

FLUENT Solid Materials

allows you to choose the solid material for which you want to modify properties. This option is available when **solid** is selected in the **Material Type** drop-down list.

FLUENT Mixture Materials

allows you to choose the mixture material for which you want to modify properties. This option is available when **mixture** is selected in the **Material Type** drop-down list.

FLUENT Droplet Particle Materials

allows you to choose the droplet-particle for which you want to modify properties. This option is available when **droplet-particle** is selected in the **Material Type** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Order Materials by

allows you to order the materials in the **Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

FLUENT Database...

opens the *FLUENT Database Materials Dialog Box* (p. 1744), from where you can copy materials from the global database into the current solver.

User-Defined Database...

opens the *Open Database Dialog Box* (p. 1746), where you can specify the user-defined database to be used.

Properties

contains input fields for the material properties that are required for the active physical models.

Density

sets the material density. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See *Density* (p. 403) for instructions on setting density. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Cp (Specific Heat)

sets the constant-pressure specific heat of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See *Specific Heat Capacity* (p. 433) for instructions on setting specific heat. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Thermal Conductivity

sets the thermal conductivity of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See *Thermal Conductivity* (p. 418) for instructions on setting thermal conductivity. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Viscosity

sets the viscosity of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See *Viscosity* (p. 408) for instructions on setting viscosity. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Molecular Weight

sets the molecular weight of the material. It is used to derive the gas constant of the material. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard State Enthalpy

specifies the formation enthalpy of a fluid material for a reacting flow. See *Standard State Enthalpies* (p. 447) for details.

Standard State Entropy

specifies the standard state entropy of a fluid material for a reacting flow. This input is used only if the fluid material is involved in a reversible reaction. See *Standard State Entropies* (p. 447) for details.

Reference Temperature

specifies the reference temperature for the **Heat of Formation**.

L-J Characteristic Length

specifies the kinetic theory parameter σ for a fluid material. See *Kinetic Theory Parameters* (p. 448) for details.

L-J Energy Parameter

specifies the kinetic theory parameter ε/k for a fluid material. See *Kinetic Theory Parameters* (p. 448) for details.

Absorption Coefficient

specifies the absorption coefficient a for radiation heat transfer. See *Radiation Properties* (p. 435) for details. If you choose the **wsggm-user-specified** option from the drop-down list next to **Absorption Coefficient**, the *WSGGM User Specified Dialog Box* (p. 1776) will open. If you choose the **user-defined-wsggm** option from the drop-down list next to **Absorption Coefficient**, the *User-Defined Functions Dialog Box* (p. 1753) will open.

Scattering Coefficient

specifies the scattering coefficient σ_s for radiation heat transfer (only for the P-1, Rosseland, or DO radiation model). See *Radiation Properties* (p. 435) for details.

Scattering Phase Function

specifies an **isotropic** (by default) or **linear-anisotropic** scattering function. If you are using the DO model, **delta-eddington** and **user-defined** scattering functions are also available. See *Radiation Properties* (p. 435) for details. If you choose **delta-eddington**, the *Delta-Eddington Scattering Function Dialog Box* (p. 1777) will open.

Refractive Index

specifies the refractive index for the material. It is used only when semi-transparent media are modeled with the DO radiation model.

Mixture Species

specifies the names of the species that comprise a mixture material. To check or modify these names, click on the **Edit...** button to open the *Species Dialog Box* (p. 1763). This property appears only for mixture materials.

Reaction

displays the reaction mechanism being used when you are modeling finite-rate reactions. **finite-rate** appears if **LaminarFinite-Rate**or **Eddy-Dissipation Concept** is selected in the *Species Model Dialog Box* (p. 1681), **eddy-dissipation** appears if **Eddy-Dissipation** is selected, and **finite-rate/eddy-dissipation** appears if **Finite-Rate/Eddy-Dissipation** is selected.

Click **Edit...** to open the *Reactions Dialog Box* (p. 1765).

Mechanism

allows you to enable different reactions selectively in different geometrical zones. Click the **Edit** button to open the *Reaction Mechanisms Dialog Box* (p. 1771). See *Defining Zone-Based Reaction Mechanisms* (p. 816) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass Diffusivity

contains a drop-down list of available methods for specifying the diffusion coefficients for the species in a mixture material. If you select **constant-dilute-appx**, you will enter a constant value in the field below. If you select **dilute-approx** or **multicomponent**, the *Mass Diffusion Coefficients Dialog Box* (p. 1773) will open, and you can specify the coefficients there. If you select **kinetic-theory**, you will need to specify the kinetic theory parameters for the individual fluid materials (species) that comprise the mixture. See *Mass Diffusion Coefficients* (p. 438) for details about specifying mass diffusivity. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Thermal Diffusion Coefficient

contains a drop-down list of available methods for specifying the thermal diffusion coefficients for the species in a mixture material. If you select **kinetic-theory**, you will need to specify the kinetic theory materials (species) that comprise the mixture. If you select **specified**, the *Thermal Diffusion Coefficients Dialog Box* (p. 1774) will open, and you can specify the coefficients there. See *Thermal Diffusion Coefficient Inputs* (p. 445) for details about specifying thermal diffusion coefficients.

Density of Unburnt Reactants

sets the density (ρ_u in Equation 9–67 in the Theory Guide) of the unburnt products.

Temperature of Unburnt Reactants

sets the temperature (T_u in Equation 9–67 in the Theory Guide) of the unburnt products.

Adiabatic Temperature of Burnt Products

(only for adiabatic premixed combustion models) specifies the value of the burnt products under adiabatic conditions, T_{ad} in Equation 9–64 in the Theory Guide.

Molecular Heat Transfer Coefficient

specifies the molecular heat transfer coefficient (α in Equation 9–9 in the Theory Guide) for use with the premixed combustion model. See *Modeling Premixed Combustion* (p. 893) for details.

Laminar Flame Speed

specifies the value of U_l in Equation 9–9 in the Theory Guide.

Laminar Flame Thickness

specifies the thickness of the flame for which you have a choice of **constant**, **user-defined**, or **diffusivity-over-flame-speed**.

Critical Rate of Strain

specifies the value of g_{cr} in Equation 9–17 in the Theory Guide.

Heat of Combustion

(only for non-adiabatic premixed combustion models) specifies the value of H_{comb} in Equation 9–66 in the Theory Guide.

Unburnt Fuel Mass Fraction

(only for non-adiabatic premixed combustion models) specifies the value of Y_{fuel} in Equation 9–66 in the Theory Guide.

Thermal Expansion Coefficient

specifies the thermal expansion coefficient (β in *Equation 14–2* (p. 697)) for use with the Boussinesq approximation. See *Steps in Solving Buoyancy-Driven Flow Problems* (p. 698) for details.

Droplet Surface Tension

specifies the value of the droplet surface tension (σ in Equation 16–226 in the Theory Guide).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Latent Heat

is the latent heat of vaporization, h_{fg}, required for phase change from an evaporating liquid droplet or for the evolution of volatiles from a combusting particle. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Latent Heat at NBP

is the latent heat of vaporization at the normal boiling point, available for droplet-particles. See *Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models* (p. 465) for details.

Normal Boiling Point (NBP)

is given by *Equation 8–105* (p. 467). See *Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models* (p. 465) for details.

Thermophoretic Coefficient

specifies the thermophoretic coefficient ($D_{T,p}$ in Equation 16–8 in the Theory Guide), and appears when the thermophoretic force is included in the discrete phase calculation.

Vaporization Temperature

is the temperature, T_{vap}, at which the calculation of vaporization from a liquid droplet or devolatilization from a combusting particle is initiated by ANSYS FLUENT. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Boiling Point

is the temperature, T_{bp}, at which the calculation of the boiling rate equation is initiated by ANSYS FLUENT. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Vapor-Particle-Equilibrium

is the selected approach for the calculation of the vapor concentration of the components at the surface. This can be Raoult's law (Equation 16–159 in the Theory Guide), the Peng-Robinson real gas model (Equation 16–168 in the Theory Guide), or a user-defined function that provides this value.

Volatile Component Fraction

(f_{v0}) is the fraction of a droplet particle that may vaporize via Laws 2 and/or 3 (Droplet Vaporization (Law 2) in the Theory Guide). For combusting particles, it is the fraction of volatiles that may be evolved via Law 4 (Devolatilization (Law 4) in the Theory Guide). See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Binary Diffusivity

is the mass diffusion coefficient, $D_{i,m}$, used in the vaporization law, Law 2. This input is also used to define the mass diffusion of the oxidizing species to the surface of a combusting particle, $D_{i,m}$. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Saturation Vapor Pressure

is the saturated vapor pressure, P_{sat}, defined as a function of temperature, which is used in the vaporization law, Law 2. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Heat of Pyrolysis

is the heat of the instantaneous pyrolysis reaction, h_{pyrol}, that the evaporating/boiling species may undergo when released to the continuous phase. The heat of pyrolysis should be input as a positive number for exothermic reaction and as a negative number for endothermic reaction. The default value of zero implies that the heat of pyrolysis is not considered. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

specifies the kinetic theory parameter f, which is the number of nodes of energy storage. This parameter is required only if you are defining specific heat via kinetic theory. See *Kinetic Theory Parameters* (p. 448) for details.

Particle Emissivity

is the emissivity of particles in your model, ε_p, used to compute radiation heat transfer to the particles when the P-1 or DO radiation model is active and particle radiation interaction is enabled in the *Discrete Phase Model Dialog Box* (p. 1723). See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Particle Scattering Factor

is the scattering factor, f, due to particles in the P-1 or DO radiation model. Note that this property will appear only if particle radiation interaction is enabled in the *Discrete Phase Model Dialog Box* (p. 1723). See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Swelling Coefficient

is the coefficient, C_{sw}, which governs the swelling of the coal particle during the devolatilization law, Law 4. A swelling coefficient of unity (the default) implies that the coal particle stays at constant diameter during the devolatilization process. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Burnout Stoichiometric Ratio

is the stoichiometric requirement, S_b, for the burnout reaction, in terms of mass of oxidant per mass of char in the particle. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Combustible Fraction

is the mass fraction of char, f_{comb}, in the coal particle, i.e., the fraction of the initial combusting particle that will react in the surface reaction, Law 5. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

React. Heat Fraction Absorbed by Solid

is the parameter f_h, which controls the distribution of the heat of reaction between the particle and the continuous phase. The default value of zero implies that the entire heat of reaction is released to the continuous phase. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Heat of Reaction for Burnout

is the heat released by the surface char combustion reaction, Law 5. This parameter is input in terms of heat release (e.g., Joules) per unit mass of char consumed in the surface reaction. See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Devolatilization Model

defines which version of the devolatilization model, Law 4, is being used. If you want to use the default constant rate devolatilization model, retain the selection of **constant** in the drop-down list to the right of **Devolatilization Model** and input the rate constant A_0 in the field below the list.

Choose **single-rate**, **two-competing-rates**, or **cpd-model** in the drop-down list to activate one of the optional devolatilization models (the single kinetic rate model, two kinetic rates model, or CPD model, as described in Devolatilization (Law 4) in the Theory Guide).

When the single kinetic rate model (**single-rate**) is selected, the *Single Rate Devolatilization Dialog Box* (p. 1778) will appear; when the two competing rates model (**two-competing-rates**) is selected, the *Two Competing Rates Model Dialog Box* (p. 1779) will appear; and when the CPD model (**cpd-model**) is selected, the *CPD Model Dialog Box* (p. 1780) will appear.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Combustion Model

defines which version of the surface char combustion law (Law 5) is being used. If you want to use the default diffusion-limited rate model, retain the selection of **diffusion-limited** in the drop-down list. No additional inputs are necessary, because the binary diffusivity defined above will be used in Equation 16–136 in the Theory Guide.

To use the kinetics/diffusion-limited rate model for the surface combustion model, select **kinetics/diffusion-limited** in the drop-down list and enter the parameters in the resulting *Kinetics/Diffusion-Limited Combustion Model Dialog Box* (p. 1780).

To use the intrinsic model for the surface combustion model, select **intrinsic-model** in the drop-down list and enter the parameters in the resulting *Intrinsic Combustion Model Dialog Box* (p. 1781).

To use the multiple surface reactions model, select **multiple-surface-reactions** in the drop-down list.

See *Setting Material Properties for the Discrete Phase* (p. 1054) for details.

Pure Solvent Melting Heat

specifies the latent heat for the melting and solidification model (L in Equation 18–2 in the Theory Guide).

Solidus Temperature

specifies the solidus temperature for the melting and solidification model ($T_{solidus}$ in Equation 18–2 in the Theory Guide). If you are solving for species transport, the solidus temperature is $T_{solidus}$ in Equation 18–8, and you specify the method by which it is calculated: either according to the **mixing-law** (which is based on the parameters of the solutes) or a **user-defined** function.

Liquidus Temperature

specifies the liquidus temperature for the melting and solidification model ($T_{liquidus}$ in Equation 18–2 in the Theory Guide). If you are solving for species transport, the liquidus temperature is T_{liquidus} in Equation 18–9, and you specify the method by which it is calculated: either according to the **mixing-law** (which is based on the parameters of the solutes) or a **user-defined** function.

Pure Solvent Melting Temperature

specifies the melting temperature of pure solvent (T_{melt} in Equation 18–8 and Equation 18–9 in the Theory Guide) for the melting and solidification model when species transport has also been enabled. The solvent is the last species listed under of the mixture material.

Eutectic Temperature

is the lowest alloy melting temperature, which depends on the relative proportions of the mixture composition of the Eutectic species mass fractions.

Slope of Liquidus Line

specifies the slope of the liquidus surface with respect to the concentration of the solute (m_i in Equation 18–8 and Equation 18–9 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Partition Coefficient

specifies the partition coefficient with respect to the concentration of the solute fluid (K_i in Equation 18–8 and Equation 18–9 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eutectic Mass Fraction
is the mass fraction of a solute in an alloy at which the melting temperature of the alloy is the lowest possible value ($Y_{i,\mathrm{Eut}}$ in Equation 18–10in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Diffusion in Solid
specifies the rate of diffusion in the solid. Note that this option is available only for the melting and solidification model when the Lever rule is selected and the species transport is enabled.

UDS Diffusivity
specifies the diffusion coefficient for a user-defined scalar. This material property is available in the *Create/Edit Materials Dialog Box* (p. 1736) when you specify one or more user-defined scalars in the *User-Defined Scalars Dialog Box* (p. 2113). If you select **defined-per-uds**, you will need to specify the diffusion coefficient for each user-defined scalar transport equation in the *UDS Diffusion Coefficients Dialog Box* (p. 1775). This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

When you are viewing the database, additional properties may be displayed. However, after you copy the material to the local area, only the properties with relevance to the current problem will be displayed.

Change/Create
changes the properties of a locally-stored material or creates a new one in the local area. If no material with the specified **Name** exists locally, ANSYS FLUENT will create it. If you have modified the material without changing its name, ANSYS FLUENT will simply update the material with your modifications. If you have assigned a new name to the material and a material with this name already exists locally, an error will be indicated; you must then specify a different name or delete the existing material with that name before trying to save the new material.

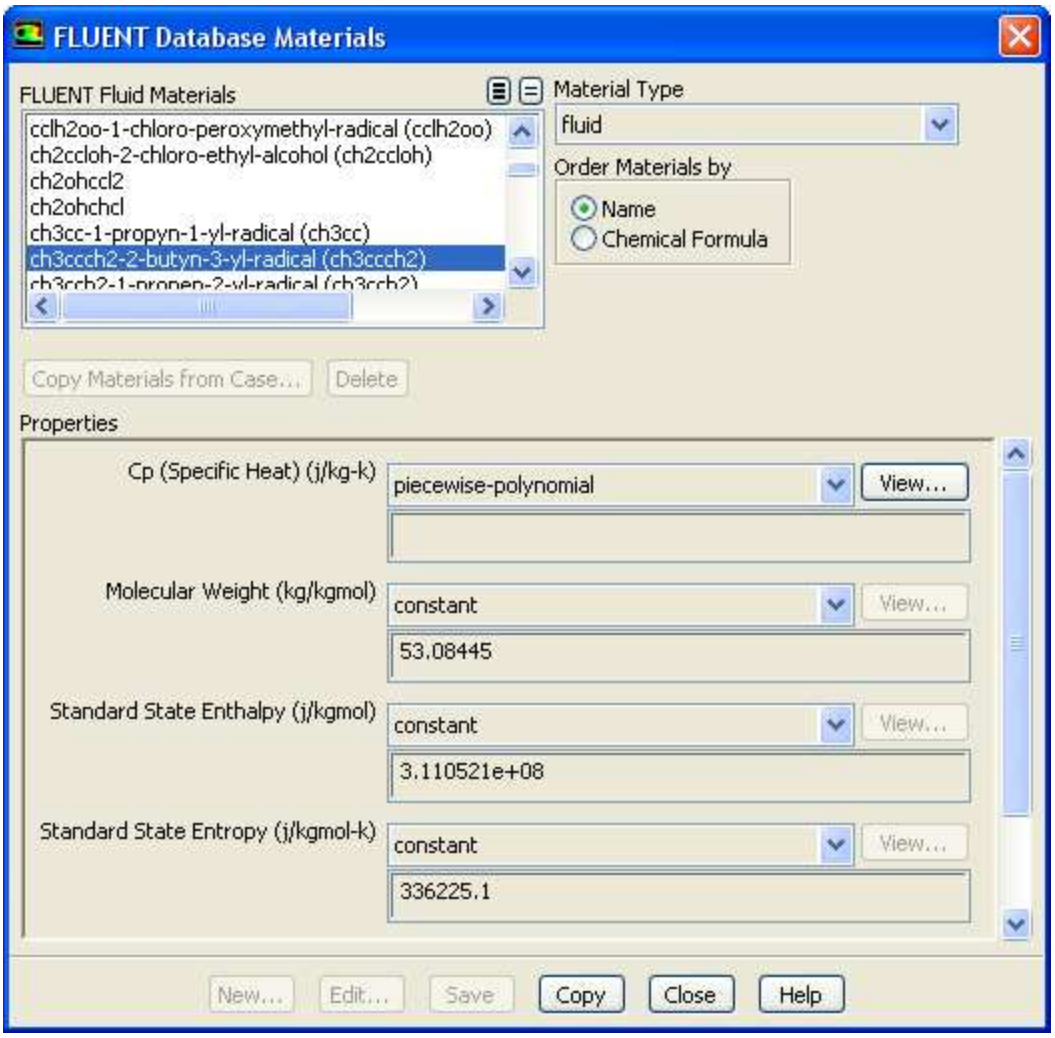
Delete
deletes the currently selected material from the local materials list. It has no effect on the global database.

35.4.2. FLUENT Database Materials Dialog Box

The **FLUENT Database Materials** dialog box is opened by clicking on the **FLUENT Database...** button in the *Create/Edit Materials Dialog Box* (p. 1736). In this dialog box you can view the global (site-wide) material properties database and copy materials list in the solver. See *Copying Materials from the ANSYS FLUENT Database* (p. 389) for details.

*There are two dialog boxes with the name **FLUENT Database Materials**. The other dialog box is opened from the Species Model Dialog Box (p. 1681).*

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

FLUENT Fluid Materials

contains a list of all materials of the selected **Material Type** that are defined in the database. The name of this list will change depending on the selected material type (e.g., **fluid**, **solid**, etc.). You can select one or more of these materials to be copied to the solver.

Material Type

is a drop-down list containing all of the available material types. By default, **fluid** and **solid** will be the only choices. If you are modeling species transport/combustion, **mixture** will also be available. For problems in which you have defined discrete-phase injections, **inert-particle**, **droplet-particle**, and/or **combusting-particle** will also appear.

Order Materials by

allows you to order the materials in the **Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Copy Materials from Case...

opens the *Copy Case Material Dialog Box* (p. 1748).

Delete

deletes the selected materials from the database.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties

contains fields for the material properties that are defined for the selected material. These fields are for informational purposes only; they cannot be edited.

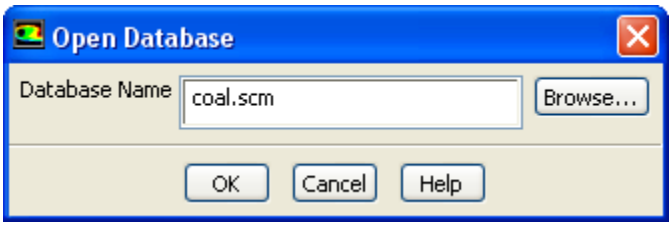
When you are viewing the database, not all properties displayed are relevant to your ANSYS FLUENT solution. After you copy the material, only properties with relevance to the current physical models will be displayed.

Copy

copies the current material from the global database to the local materials list in the solver.

35.4.3. Open Database Dialog Box

The **Open Database** dialog box is opened by clicking on the **User-Defined Database** button in the *Create/Edit Materials Dialog Box* (p. 1736).



Controls

Browse...

opens the *The **Select File** Dialog Box* (p. 33) where you can select the user-defined database to be used in the current solver session.

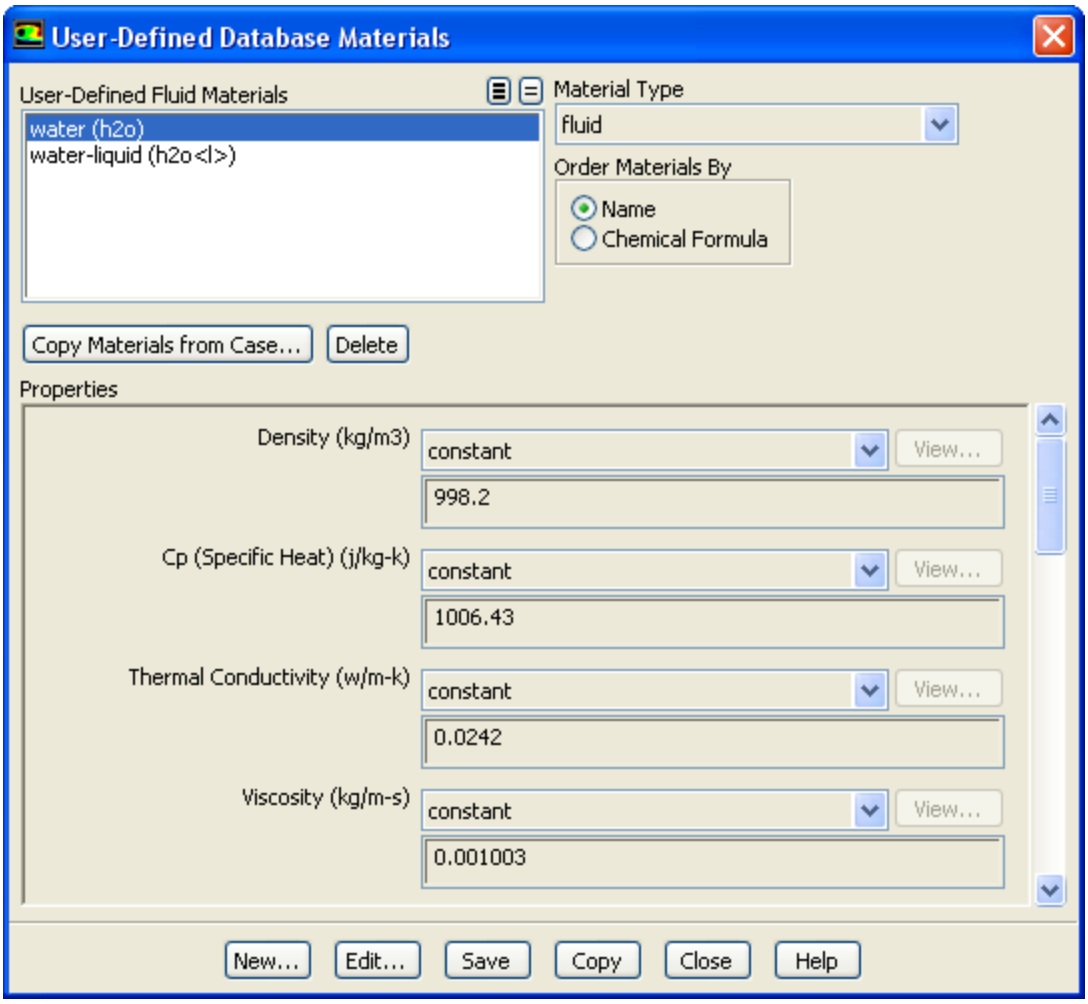
Database Name

allows to enter the path and name of a new database. If you select an existing database, this filed displays the path and name of the selected database.

35.4.4. User-Defined Database Materials Dialog Box

The **User-Defined Database Materials** dialog box is opened by clicking **OK** in the *Open Database Dialog Box* (p. 1746). In this dialog box you can view the user-defined material properties database and copy materials list in the solver. See *Viewing Materials in a User-Defined Database* (p. 392) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

User-Defined Materials

contains a list of all materials of the selected **Material Type** that are defined in the database. The name of this list will change depending on the selected material type (e.g., **User-Defined Fluid Materials**, **User-Defined Solid Materials**, etc.). You can select one or more of these materials to copy to your local list or edit their properties.

Material Type

is a drop-down list containing all of the available material types.

Order Materials By

allows you to order the materials in the **User-Defined Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Copy Materials from Case...

opens the *Copy Case Material Dialog Box* (p. 1748).

Delete

deletes the selected materials from the database.

Properties

lists the properties and values of the selected material.

New...

opens a blank *Material Properties Dialog Box* (p. 1748) where you can define a new material.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

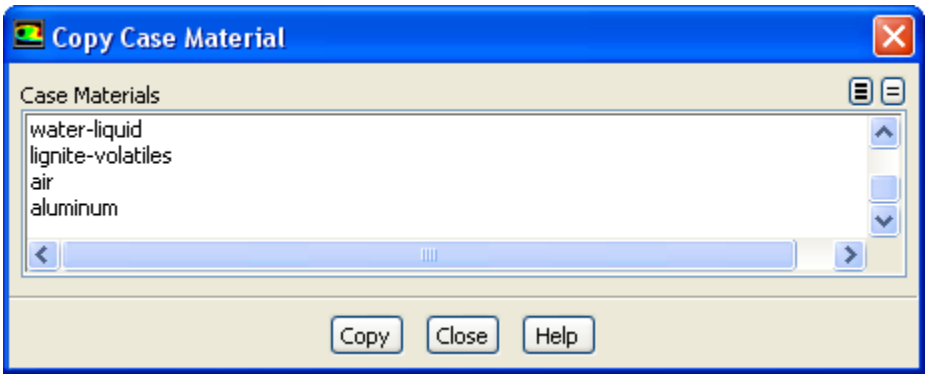
Edit...
opens the *Material Properties Dialog Box* (p. 1748) displaying the properties of the selected material.

Save
saves the information to the selected database.

Copy
copies the selected material to your local material list. If the material already exists the *New Material Name Dialog Box* (p. 1750) opens

35.4.5. Copy Case Material Dialog Box

This dialog box is opened by clicking the **Copy Materials from Case...** button in the *User-Defined Database Materials Dialog Box* (p. 1746).



Controls

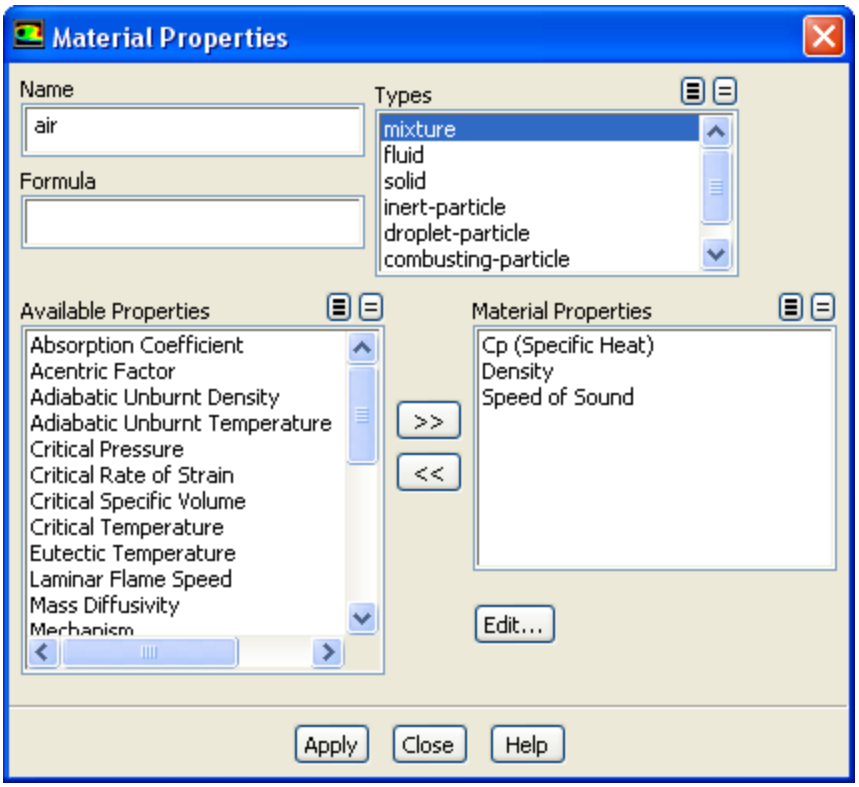
Case Materials
lists all the materials present in your local materials list.

Copy
copies the selected materials to the user-defined database.

35.4.6. Material Properties Dialog Box

This dialog box is opened by clicking **New...** button in the *User-Defined Database Materials Dialog Box* (p. 1746). See *Creating a New Materials Database and Materials* (p. 395) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
specifies the name of the material that you are creating.

Formula
(optional) specifies the chemical formula of the material that you are creating.

Types
allows you to select material type from fluid, solid, inter-particle, droplet-particle, combusting-particle, and mixture materials.

Available Properties
lists properties applicable to the selected material type.

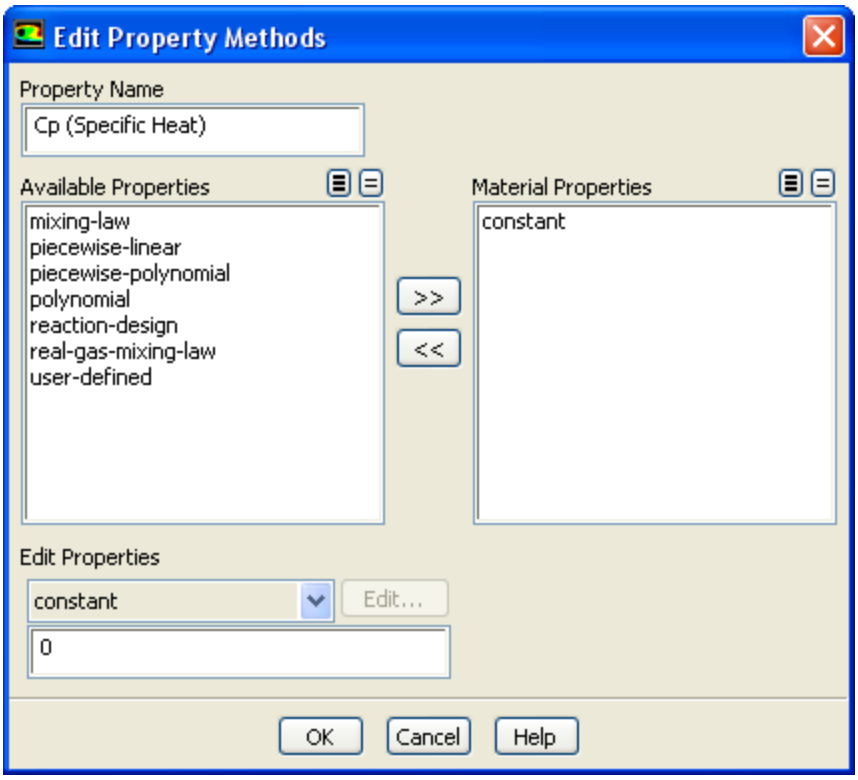
Material Properties
lists properties that you have selected from **Available Properties** list.

Edit...
opens *Edit Property Methods Dialog Box* (p. 1749) where you can edit the parameters that define a property.

35.4.7. Edit Property Methods Dialog Box

This dialog box is opened by clicking **Edit...** button in the *Material Properties Dialog Box* (p. 1748). See *Creating a New Materials Database and Materials* (p. 395) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Property Name
specifies the name of the property that you want to edit.

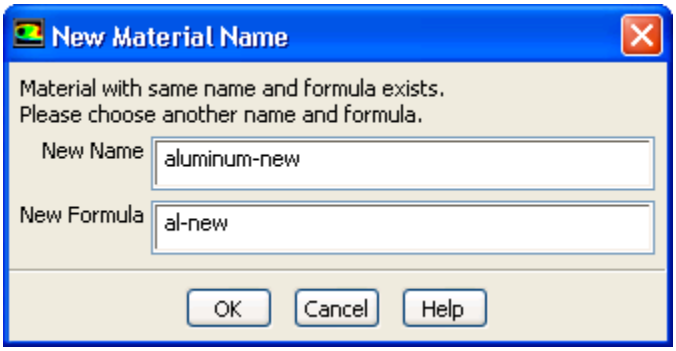
Available Properties
specifies the methods that can be used to define the selected property.

Material Properties
lists properties that you have selected from **Available Properties** list.

Edit Properties
allows you to select the property that you want to edit.

35.4.8. New Material Name Dialog Box

This dialog box is opened by clicking the **Copy** button in the *User-Defined Database Materials Dialog Box* (p. 1746) when a material is already defined with the same name in the *Create/Edit Materials Dialog Box* (p. 1736).



Controls

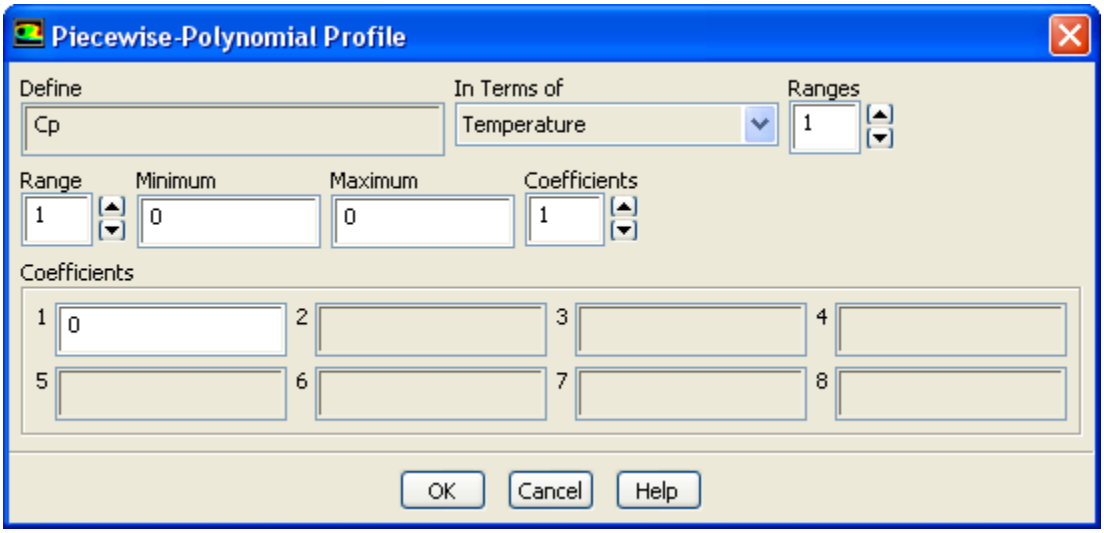
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Name
allows you to enter the new name for the material you need to copy.

New Formula
allows you to enter the new formula for the material you need to copy.

35.4.9. Polynomial Profile Dialog Box

The **Polynomial Profile** dialog box allows you to define a physical property as a polynomial function of temperature. This dialog box will open when you select **polynomial** in the drop-down list next to a physical property in the *Create/Edit Materials Dialog Box* (p. 1736). See *Inputs for Polynomial Functions* (p. 399) for details about the items below.



Controls

Define
shows the property that is being defined as a function of temperature.

In Terms of
shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a polynomial function of temperature.

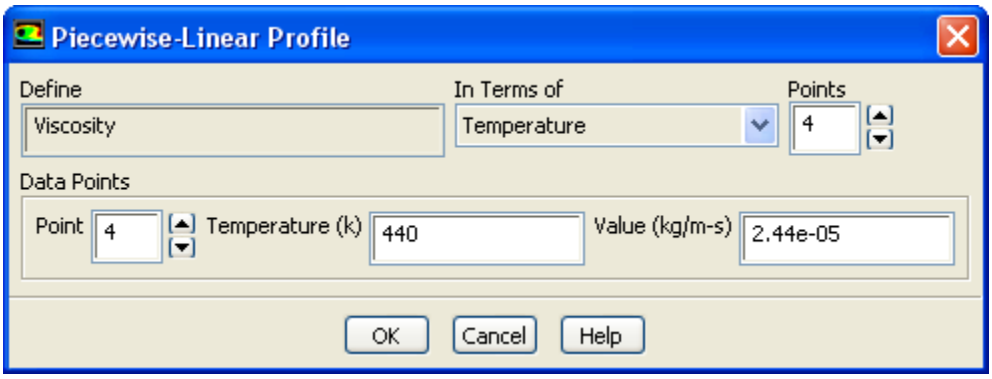
Coefficients
is an integer number entry that indicates the number of coefficients to be defined. You can define up to 8 coefficients.

Coefficients
contains real number entries for the number of coefficients set in the **Coefficients** integer number entry above. The number of entries that are editable will be the same as the number of coefficients you requested.

35.4.10. Piecewise-Linear Profile Dialog Box

The **Piecewise-Linear Profile** dialog box allows you to define a physical property as a piecewise-linear function of temperature. This dialog box will open when you select **piecewise-linear** in the drop-down list next to a physical property in the *Create/Edit Materials Dialog Box* (p. 1736). See *Inputs for Piecewise-Linear Functions* (p. 400) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Define
shows the property that is being defined as a function of temperature.

In Terms of
shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a piecewise-linear function of temperature.

Points
indicates the number of data pairs that will define the piecewise distribution. You can define up to 30 pairs.

Data Points
contains entries for defining the data pairs.

Point
indicates the point for which the data pair (**Temperature**, **Value**) is being defined.

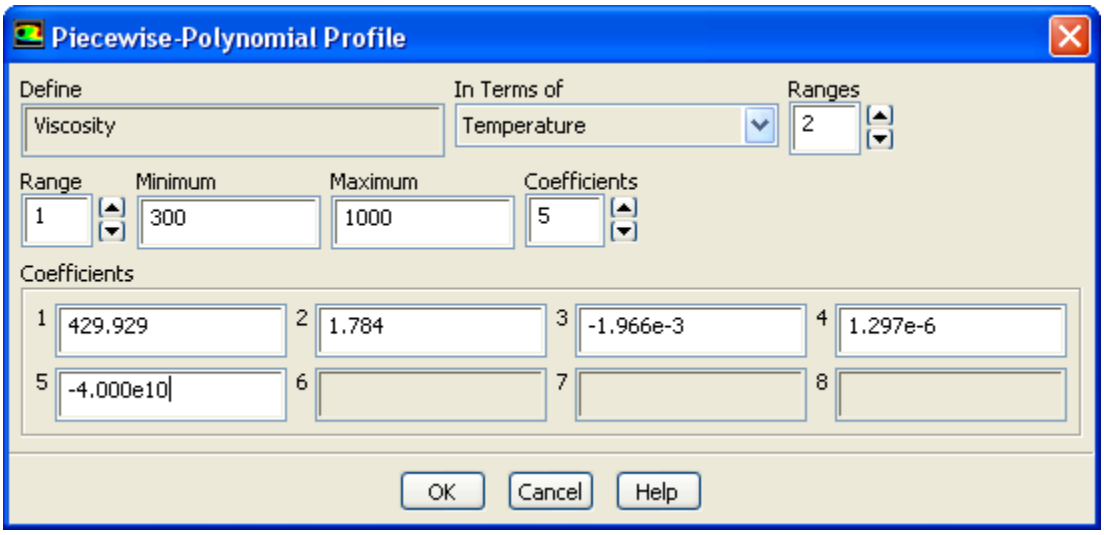
Temperature
is the independent variable.

Value
is the dependent variable (i.e., the property). In the example dialog box above, **Viscosity** is the variable being defined, as shown in the **Define** field.

35.4.11. Piecewise-Polynomial Profile Dialog Box

The **Piecewise-Polynomial Profile** dialog box allows you to define a physical property as a piecewise-polynomial function of temperature. This dialog box will open when you select **piecewise-polynomial** in the drop-down list next to a physical property in the *Create/Edit Materials Dialog Box* (p. 1736). See *Inputs for Piecewise-Polynomial Functions* (p. 401) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Define
shows the property that is being defined as a function of temperature.

In Terms of
shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a polynomial function of temperature.

Ranges
sets the number of temperature ranges for which you will define polynomial functions. You can define up to 3 ranges.

Range
indicates the temperature range for which you are defining the polynomial function.

Minimum, Maximum
set the minimum and maximum temperatures for the specified **Range**.

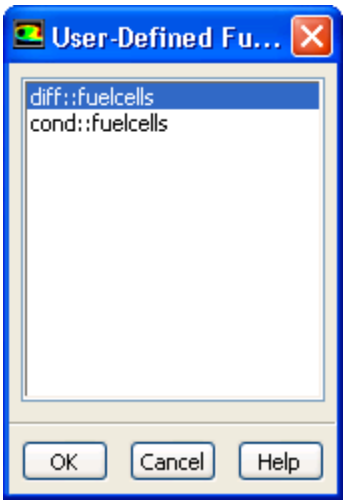
Coefficients
is an integer number entry that indicates the number of coefficients to be defined for the specified **Range**. You can define up to 8 coefficients.

Coefficients
contains real number entries for the number of coefficients set in the **Coefficients** integer number entry above. The number of entries that are editable will be the same as the number of coefficients you requested for the specified **Range**.

35.4.12. User-Defined Functions Dialog Box

The **User-Defined Functions** dialog box allows you to choose which user-defined function is to be used to define a material property. This dialog box will open when you select **user-defined** in the drop-down list next to one of the **Properties** in the *Create/Edit Materials Dialog Box* (p. 1736). See the separate UDF Manual for details about user-defined functions.

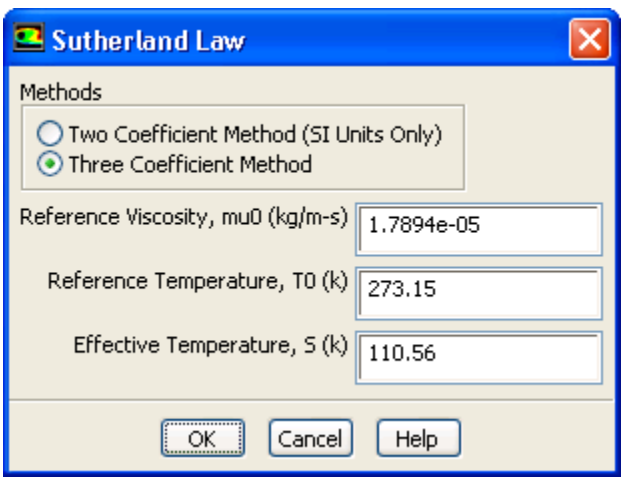
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The list will contain all available user-defined functions.

35.4.13. Sutherland Law Dialog Box

The **Sutherland Law** dialog box allows you to set the coefficients for Sutherland's law for viscosity. This dialog box will open when you select **sutherland** in the drop-down list next to **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Sutherland Viscosity Law* (p. 410) for details about the items below.



Controls

Methods

contains options for selecting the **Two Coefficient Method** or the **Three Coefficient Method**.

C1, C2

set the coefficients C_1 and C_2 in *Equation 8–17* (p. 410) in SI units. These inputs will appear if you select the **Two Coefficient Method**.

Reference Viscosity

sets the reference viscosity μ_0 in *Equation 8–18* (p. 410). This input will appear if you select the **Three Coefficient Method**.

Reference Temperature

sets the reference temperature T_0 in *Equation 8–18* (p. 410). This input will appear if you select the **Three Coefficient Method**.

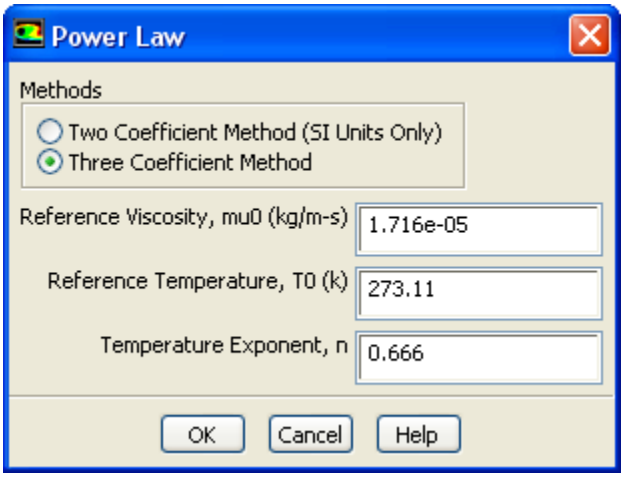
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Effective Temperature

sets the effective temperature S in *Equation 8–18* (p. 410). This input will appear if you select the **Three Coefficient Method**.

35.4.14. Power Law Dialog Box

The **Power Law** dialog box allows you to set the coefficients for the power law for viscosity. This dialog box will open when you select **power-law** in the drop-down list next to **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Power-Law Viscosity Law* (p. 411) for details about the items below.



Controls

Methods

contains options for selecting the **Two Coefficient Method** or the **Three Coefficient Method**.

B

sets the coefficient B in *Equation 8–19* (p. 411) in SI units. This input will appear if you select the **Two Coefficient Method**.

Reference Viscosity

sets the reference viscosity μ_0 in *Equation 8–20* (p. 411). This input will appear if you select the **Three Coefficient Method**.

Reference Temperature

sets the reference temperature T_0 in *Equation 8–20* (p. 411). This input will appear if you select the **Three Coefficient Method**.

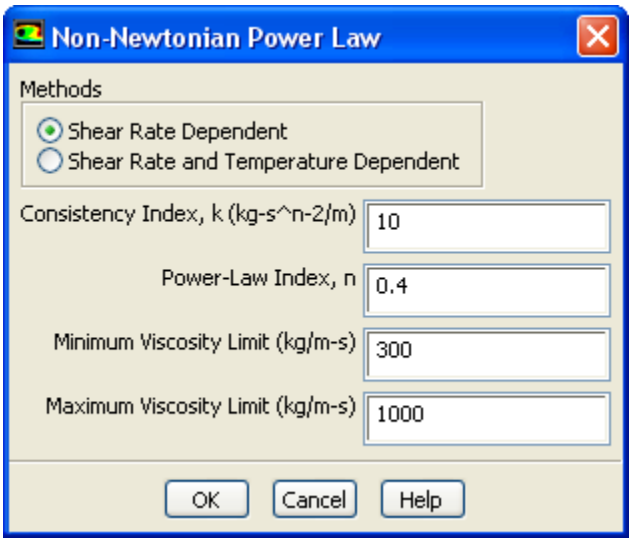
Temperature Exponent

sets the temperature exponent n in *Equation 8–19* (p. 411) or *Equation 8–20* (p. 411), depending on your **Method** selection. If you are using the **Two Coefficient Method**, this input must be in SI units.

35.4.15. Non-Newtonian Power Law Dialog Box

The **Non-Newtonian Power Law** dialog box allows you to set the parameters for the non-Newtonian power law for viscosity. This dialog box will open when you select **non-newtonian-power-law** in the drop-down list next to **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Power Law for Non-Newtonian Viscosity* (p. 415) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods
allows you to select the type of dependency on the viscosity.

Shear Rate Dependent
is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent
is where the viscosity is dependent on the shear rate and the temperature.

Consistency Index
sets the consistency index k in *Equation 8–32* (p. 415).

Power-Law Index
sets the power-law index n in *Equation 8–32* (p. 415).

Minimum Viscosity Limit, Maximum Viscosity Limit
set the minimum and maximum viscosity limits.

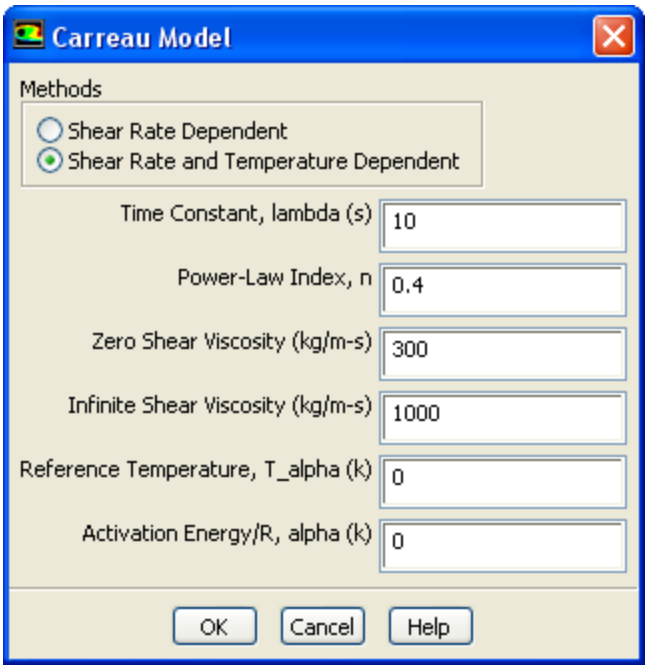
Reference Temperature
sets the reference temperature.

Activation Energy/R, alpha
is the ratio of the activation energy to the thermodynamic constant α in *Equation 8–31* (p. 414).

35.4.16. Carreau Model Dialog Box

The **Carreau Model** dialog box allows you to set the parameters for the non-Newtonian Carreau model for viscosity. This dialog box will open when you select **carreau** in the drop-down list next to **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *The Carreau Model for Pseudo-Plastics* (p. 415) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods

allows you to select the type of dependency on the viscosity.

Shear Rate Dependent

is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent

is where the viscosity is dependent on the shear rate and the temperature.

Time Constant, lambda

sets the time constant λ in *Equation 8–33* (p. 415).

Power-Law Index

sets the power-law index n in *Equation 8–33* (p. 415).

Zero Shear Viscosity, Infinite Shear Viscosity

set the zero and infinite shear viscosity limits η_0 and η_∞ in *Equation 8–33* (p. 415).

Reference Temperature, T_alpha

sets the reference temperature T_0 in *Equation 8–33* (p. 415).

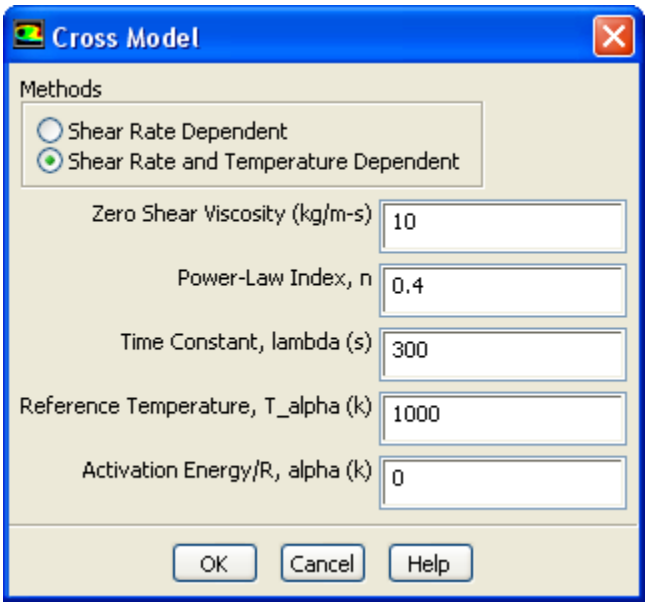
Activation Energy/R, alpha

is the ratio of the activation energy to the thermodynamic constant α in *Equation 8–31* (p. 414).

35.4.17. Cross Model Dialog Box

The **Cross Model** dialog box allows you to set the parameters for the non-Newtonian Cross model for viscosity. This dialog box will open when you select **cross** in the drop-down list next to **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Cross Model* (p. 416) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods
allows you to select the type of dependency on the viscosity.

Shear Rate Dependent
is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent
is where the viscosity is dependent on the shear rate and the temperature.

Zero Shear Viscosity
sets the zero shear viscosity limit η_0 in *Equation 8–34* (p. 416).

Power-Law Index
sets the power-law index n in *Equation 8–34* (p. 416).

Time Constant
sets the time constant λ in *Equation 8–34* (p. 416).

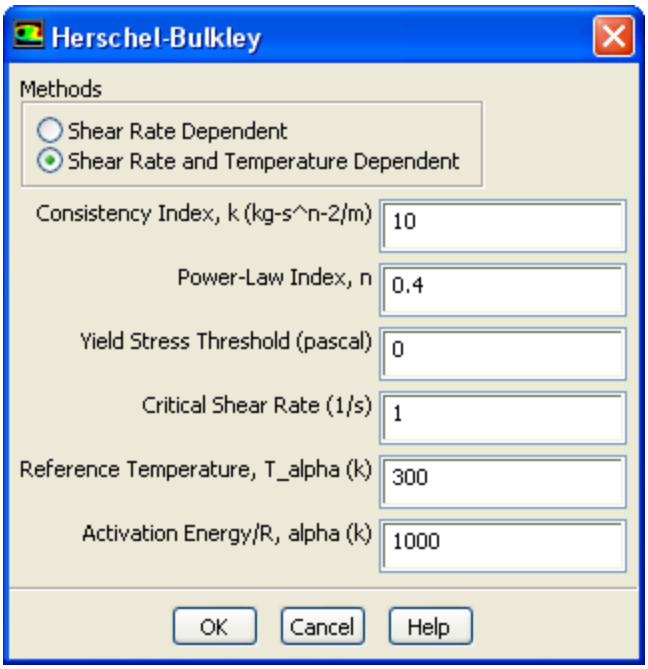
Reference Temperature, T_alpha
sets the reference temperature T_0 in *Equation 8–33* (p. 415).

Activation Energy/R, alpha
is the ratio of the activation energy to the thermodynamic constant α in *Equation 8–31* (p. 414).

35.4.18. Herschel-Bulkley Dialog Box

The **Herschel-Bulkley** dialog box allows you to set the parameters for the non-Newtonian Herschel-Bulkley model for viscosity. This dialog box will open when you select **herschel-bulkley** in the drop-down list next to **Viscosity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Herschel-Bulkley Model for Bingham Plastics* (p. 417) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods

allows you to select the type of dependency on the viscosity.

Shear Rate Dependent

is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent

is where the viscosity is dependent on the shear rate and the temperature.

Consistency Index

sets the consistency index k in *Equation 8–36* (p. 417).

Power-Law Index

sets the power-law index n in *Equation 8–36* (p. 417).

Yield Stress Threshold

sets the yield stress threshold τ_0 in *Equation 8–36* (p. 417).

Critical Shear Rate

set the critical shear rate $\dot{\gamma}_c$ in *Equation 8–36* (p. 417).

Reference Temperature, T_alpha

sets the reference temperature T_0 in *Equation 8–33* (p. 415).

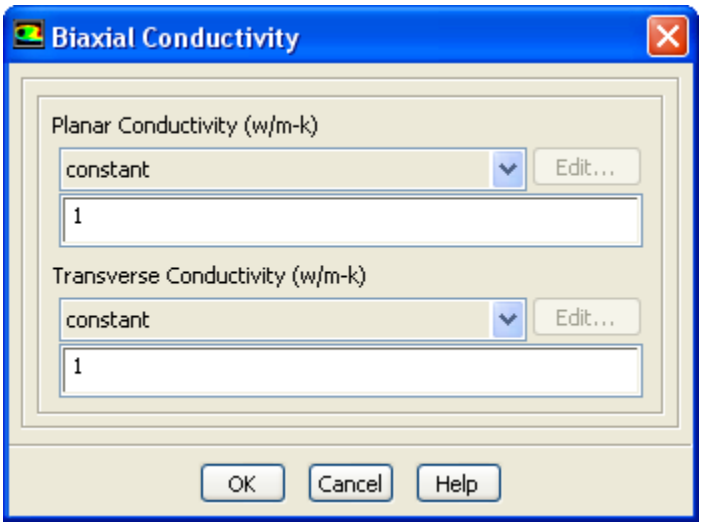
Activation Energy/R, alpha

is the ratio of the activation energy to the thermodynamic constant α in *Equation 8–31* (p. 414).

35.4.19. Biaxial Conductivity Dialog Box

The **Biaxial Conductivity** dialog box allows you to define a biaxial orthotropic thermal conductivity, which is mainly applicable to solid materials used for the wall shell conduction model. This dialog box will open when you select **biaxial** in the drop-down list next to **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Biaxial Thermal Conductivity* (p. 423) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

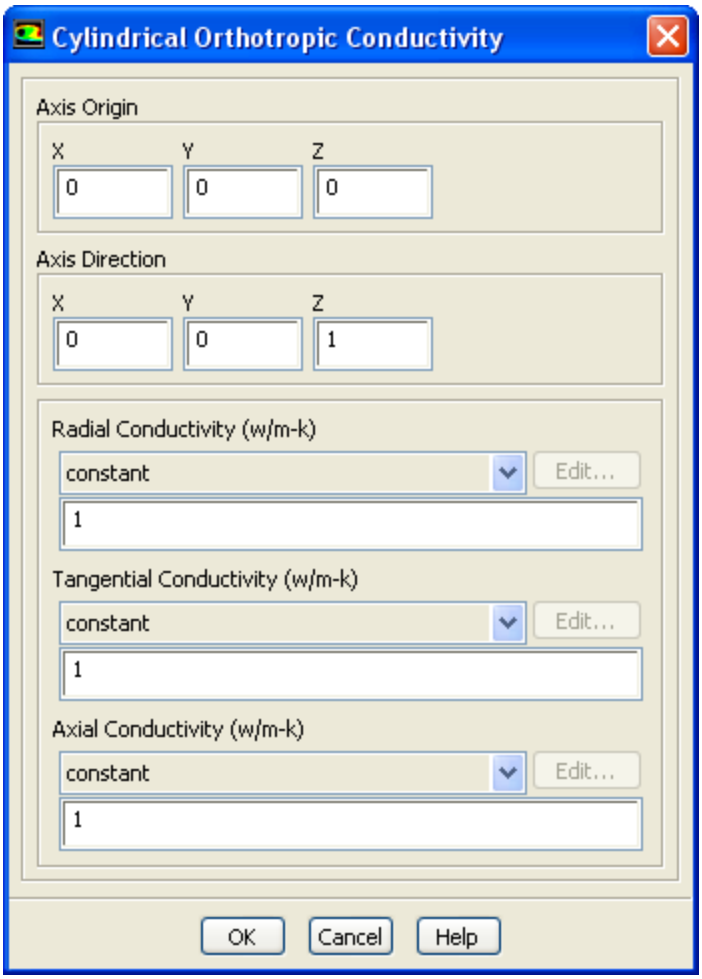
Planar Conductivity
specifies the conductivity within the shell (or solid) region.

Transverse Conductivity
specifies the conductivity normal to the surface of the solid region.

35.4.20. Cylindrical Orthotropic Conductivity Dialog Box

The **Cylindrical Orthotropic Conductivity** dialog box allows you to define an orthotropic thermal conductivity in cylindrical coordinates. This dialog box will open when you select **cyl-orthotropic** in the drop-down list next to **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Cylindrical Orthotropic Thermal Conductivity* (p. 426) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Axis Origin

allows you to specify the origin of the cylindrical coordinate system.

X, Y

specify the X, Y and Z (for three dimensional system) coordinates.

Axis Direction

(3D only) allows you to specify the direction of the axis.

X, Y

specify 1 against the direction of the axis.

Radial Conductivity

specifies the conductivity in the radial direction.

Tangential Conductivity

specifies the conductivity in the tangential direction.

Axial Conductivity

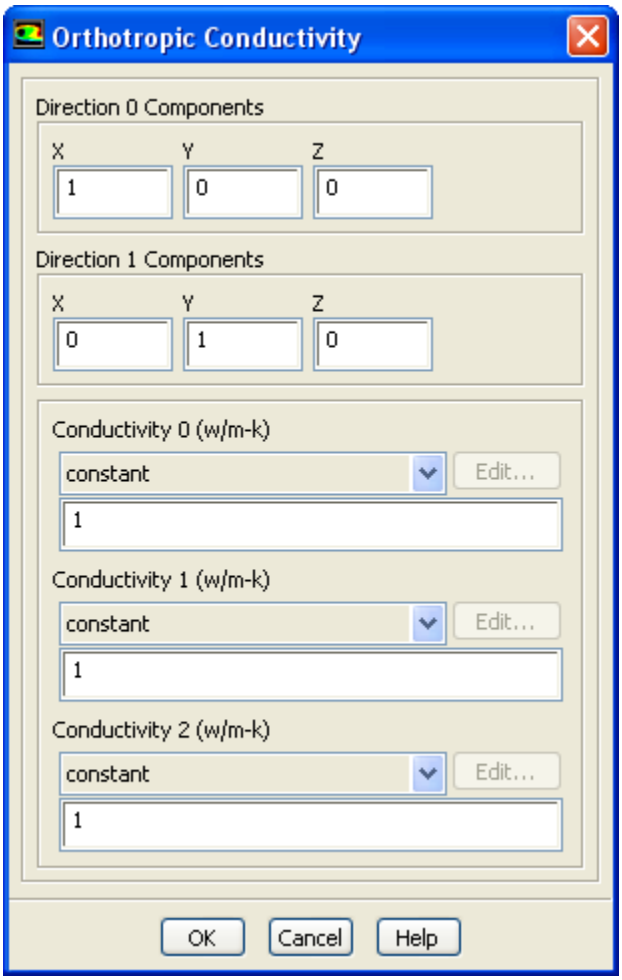
(3D only) specifies the conductivity in the axial direction.

35.4.21. Orthotropic Conductivity Dialog Box

The **Orthotropic Conductivity** dialog box allows you to define an orthotropic thermal conductivity for a solid material. This dialog box will open when you select **orthotropic** in the drop-down list next to **Thermal**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Conductivity in the *Create/Edit Materials Dialog Box* (p. 1736). See *Orthotropic Thermal Conductivity* (p. 424) for details about the items below.



Controls

Direction 0 Components, Direction 1 Components

specify the directions $\hat{e}_\xi$ and $\hat{e}_\eta$ in *Equation 8–47* (p. 424) as **X,Y,Z** vectors. For 2D cases, only **Direction 0 Components** will appear.

Conductivity 0, Conductivity 1, Conductivity 2

specify (k_ξ), (k_η), and (k_ζ) in *Equation 8–47* (p. 424) as **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** functions of temperature. For 2D cases, only **Conductivity 0** and **Conductivity 1** will appear.

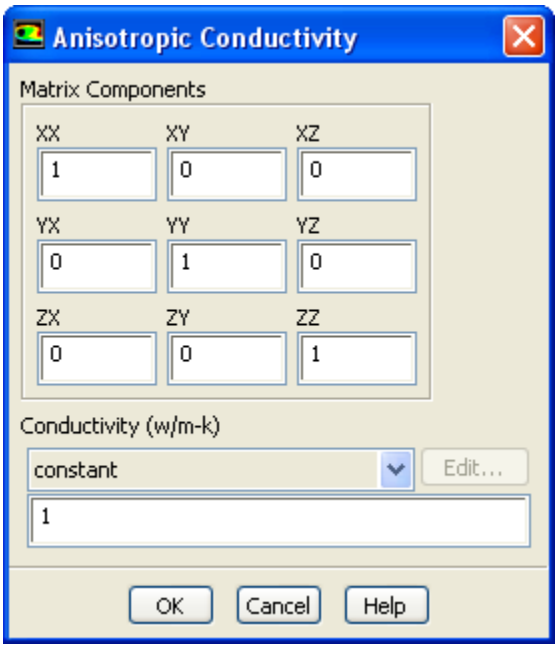
Edit...

opens the appropriate dialog box for input of a temperature-dependent conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

35.4.22. Anisotropic Conductivity Dialog Box

The **Anisotropic Conductivity** dialog box allows you to define a general anisotropic thermal conductivity. This dialog box will open when you select **anisotropic** in the drop-down list next to **Thermal Conductivity** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Anisotropic Thermal Conductivity* (p. 422) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Matrix Components

specify the components of the matrix $\hat{e}_{ij}$ in *Equation 8–46* (p. 422).

Conductivity

specifies the value of k in *Equation 8–46* (p. 422) as a **constant**, or as a **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of temperature.

Edit...

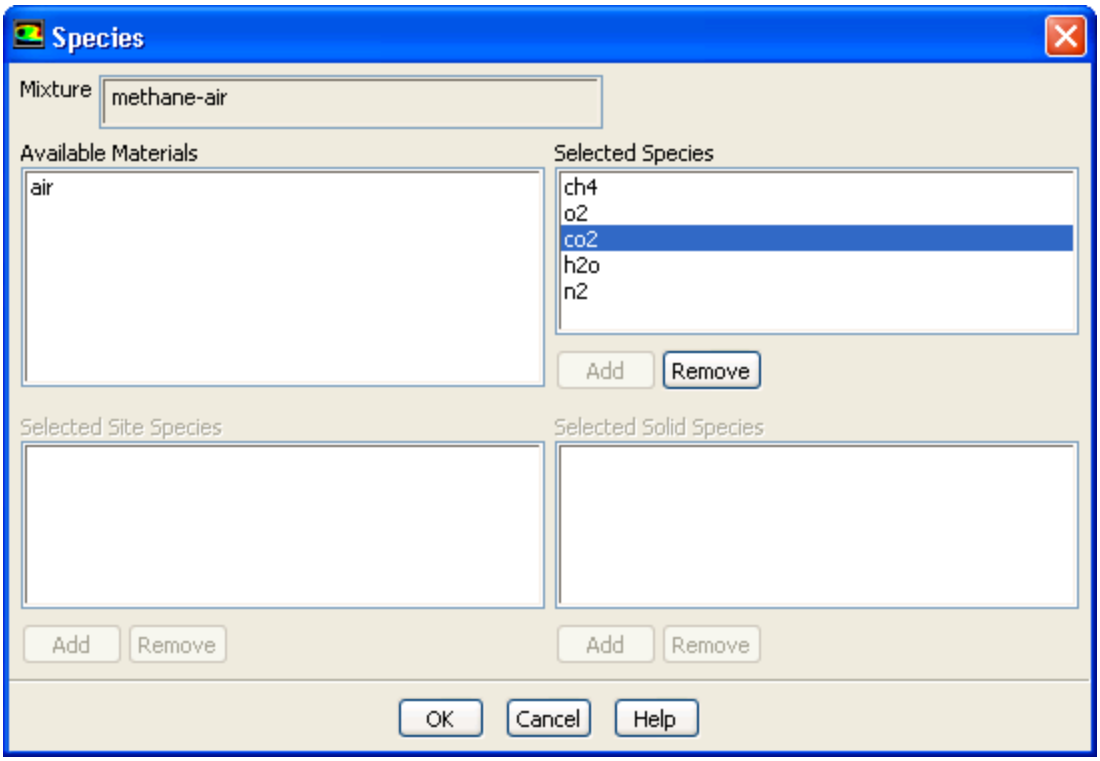
opens the appropriate dialog box for input of a temperature-dependent conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

35.4.23. Species Dialog Box

The **Species** dialog box (opened by clicking on the **Edit...** button next to **Mixture Species** in the *Create/Edit Materials Dialog Box* (p. 1736))) allows you to define the species that comprise a mixture material. See *Defining the Species in the Mixture* (p. 806) for details about the items below.

(Note that the **Species** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mixture

shows the name of the mixture material for which you are defining the species. This field is not editable.

Available Materials

is a list of all of the materials to be a component in the mixture material by selecting it and clicking on the **Add** button below the **Selected Species** or **Selected Surface Species** list. To add a material to the **Available Materials** list, use the *Fluent Database Materials Dialog Box* (p. 1783) to copy the fluid material to local storage.

Selected Species

is a list of all the fluid-phase species in the mixture. To add a material to the list, select it in the **Available Materials** list and click on the **Add** button below the **Selected Species** list. To remove a material, select it in the **Selected Species** list and click on **Remove**. See *Overview of the Species Dialog Box* (p. 807) for more information.

Selected Solid Species

is a list of all the solid species in the mixture. To add a material to the list, select it in the **Available Materials** list and click on the **Add** button below the **Selected Solid Species** list. To remove a material, select it in the **Selected Solid Species** list and click on **Remove**. See *Overview of the Species Dialog Box* (p. 807) for more information.

Selected Site Species

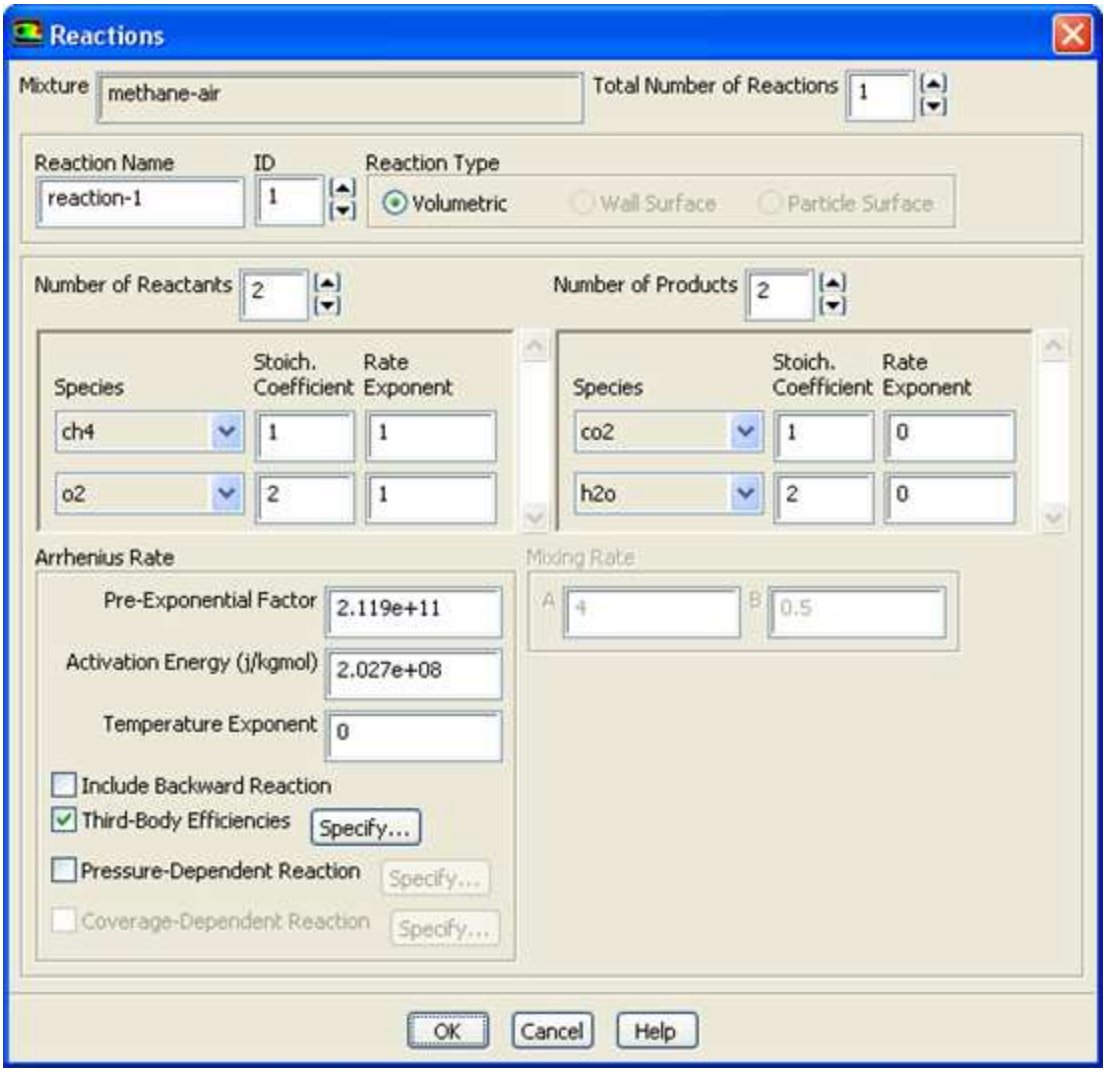
is a list of all the site species in the mixture. To add a material to the list, select it in the **Available Materials** list and click on the **Add** button below the **Selected Site Species** list. To remove a material, select it in the **Selected Site Species** list and click on **Remove**. See *Overview of the Species Dialog Box* (p. 807) for more information.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.24. Reactions Dialog Box

The **Reactions** dialog box (opened by clicking on the **Edit...** button next to **Reaction** in the *Create/Edit Materials Dialog Box* (p. 1736)) allows you to define the reactions that comprise a mixture material. See *Defining Reactions* (p. 809) for details about using this dialog box.

(Note that the **Reactions** dialog box is a modal dialog, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Mixture

shows the name of the mixture material for which you are defining the species. This field is not editable.

Total Number of Reactions

sets the total number of reactions (fluid-phase reactions and surface reactions occurring at wall boundaries). Use the arrows to change the value, or type in the value and press RETURN.

Reaction Name

contains the name of the reaction.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ID

sets the number of the reaction you want to define. (Again, if you type in the value be sure to press RETURN.)

Reaction Type

contains options that allow you to specify the type of reaction.

Volumetric

specifies, if enabled, that the current reaction is a volumetric reaction.

Wall Surface

specifies, if enabled, that the current reaction is a wall surface reaction.

Particle Surface

specifies, if enabled, that the current reaction is a particle surface reaction.

Number of Reactants

indicates the number of reactants in the specified reaction.

Species

contains drop-down lists of all species in the mixture. (The number of lists will be equal to the **Number of Reactants**.) Select each reactant in one of these lists.

Stoich. Coefficient

specifies the stoichiometric coefficient of the reactant species in the reaction.

Rate Exponent

specifies the rate constant for the reactant species in the reaction.

Number of Products

indicates the number of products in the specified reaction.

Species

contains drop-down lists of all species in the mixture. (The number of lists will be equal to the **Number of Products**.) Select each product in one of these lists.

Stoich. Coefficient

specifies the stoichiometric coefficient of the product species in the reaction.

Rate Exponent

specifies the rate constant for the product species in the reaction.

Arrhenius Rate

contains inputs related to the Arrhenius rate. (If you have chosen **Eddy-Dissipation** for the **Turbulence-Chemistry Interaction** in the *Species Model Dialog Box* (p. 1681), these inputs are not required.)

Pre-exponential Factor

is the constant A_r in Equation 7–10 in the Theory Guide. The units of A_r depend on the other rate constant inputs, but must be defined such that the units of the reaction rate $R_{i,r}$ (Equation 7–5 in the Theory Guide) are in (kg/m^3-s) if you are using SI units.

Important

It is important to note that if you have selected the British units system, the Arrhenius factor should still be input in SI units. This is because ANSYS FLUENT applies no conversion factor to your input of A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your inputs for $v_{j,r}$, β_r, etc.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Activation Energy

is the constant E_r in the forward rate constant expression, Equation 7–10 in the Theory Guide).

Temperature Exponent

is the value for the constant β_r in Equation 7–10 in the Theory Guide.

Include Backward Reaction

specifies that the reaction is reversible. The backward reaction rate constant will be computed from Equation 7–10 in the Theory Guide.

Third-Body Efficiencies

enables the input and use of third-body efficiencies ($\gamma_{j,r}$ in Equation 7–9 in the Theory Guide). These inputs are optional. (This item is available only if you have selected **Volumetric** for the **Reaction Type**.)

Pressure-Dependent Reaction

enables the modeling of a pressure fall-off reaction. See *Inputs for Reaction Definition* (p. 809) **Laminar Finite-Rate** or **Eddy-Dissipation Concept** for the **Turbulence-Chemistry Interaction** in the *Species Model Dialog Box* (p. 1681) and have selected **Volumetric** for the **Reaction Type**.)

Coverage-Dependent Reaction

is used when modeling **Wall Surface** reactions with site-balancing and reaction rates depend on site coverages.

Specify...

opens the *Third-Body Efficiencies Dialog Box* (p. 1768), the *Pressure-Dependent Reaction Dialog Box* (p. 1768), or *Coverage-Dependent Reaction Dialog Box* (p. 1770) in which you can specify the third-body efficiencies, pressure-dependent reaction parameters, or coverage parameters.

Mixing Rate

contains inputs related to the mixing rate. (If you have chosen **Laminar Finite-Rate** or **Eddy-Dissipation Concept** for the **Turbulence-Chemistry Interaction** in the *Species Model Dialog Box* (p. 1681), these inputs are not required.)

A

is the constant A in the turbulent mixing rate (Equation 7–24 and Equation 7–25 in the Theory Guide) when it is applied to a species that appears as a reactant in this reaction. The default setting of 4.0 is based on the empirically derived values given by Magnussen et al. [*49*] (p. 2211).

B

is the constant B in the turbulent mixing rate (Equation 7–25 in the Theory Guide) when it is applied to a species that appears as a product in this reaction. The default setting of 0.5 is based on the empirically derived values given by Magnussen et al. [*49*] (p. 2211).

Particle Surface Reaction

contains inputs related to a particle surface reaction. See *User Inputs for Particle Surface Reactions* (p. 835) for details. (This section will appear only if you have selected **Particle Surface** for the **Reaction Type**.)

Diffusion Limited Species

is a drop-down list that allows you to select the species for which the concentration gradient between the bulk and the particle surface is the largest when there is more than one gaseous reactant taking part in the particle surface reaction. See *User Inputs for Particle Surface Reactions* (p. 835) for details.

Diffusion Rate Constant

is the constant $C_{1,r}$ in Equation 7–68 in the Theory Guide.

Effectiveness Factor

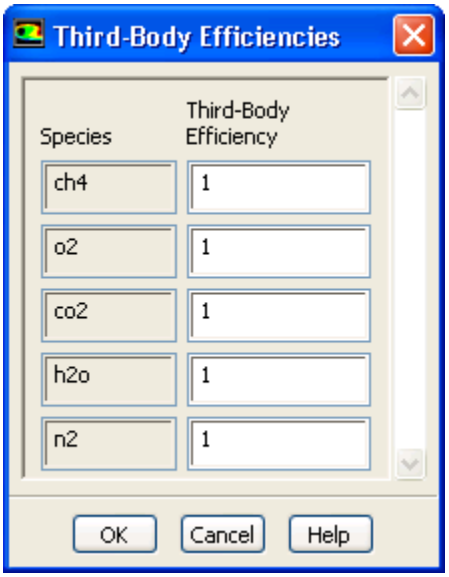
is the constant η_r in Equation 7–66 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.25. Third-Body Efficiencies Dialog Box

The **Third-Body Efficiencies** dialog box (opened by clicking on **Specify...** next to the **Third-Body Efficiencies** button in the *Reactions Dialog Box* (p. 1765)) allows you to specify the third-body efficiencies for each species in the mixture, to be used in Equation 7–9 in the Theory Guide. See *Defining Reactions* (p. 809) for details.

(Note that the **Third-Body Efficiencies** dialog box is a model dialog boxl, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Species
displays the name of each species in the mixture.

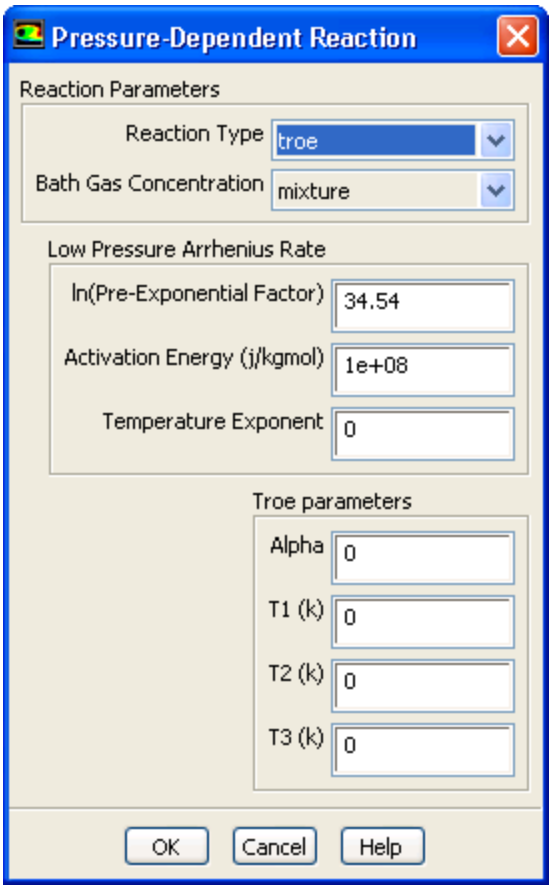
Third-body Efficiency
specifies the third-body efficiency for each species.

35.4.26. Pressure-Dependent Reaction Dialog Box

The **Pressure-Dependent Reaction** dialog box (opened by clicking on **Specify...** under **Pressure-Dependent Reaction** in the *Reactions Dialog Box* (p. 1765)) allows you to specify parameters for a pressure fall-off reaction. See *Inputs for Reaction Definition* (p. 809) for details.

(Note that the **Pressure-Dependent Reaction** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reaction Parameters

contains inputs for specifying the type of pressure fall-off reaction and the reaction parameters. See Pressure-Dependent Reactions in the Theory Guide for details.

Reaction Type

contains a drop-down list of the available reaction types: **lindemann**, **troe**, and **sri**. See Pressure-Dependent Reactions in the Theory Guide for details.

Bath Gas Concentration

allows you to specify if the bath gas concentration ($[M]$ in Equation 7–18 in the Theory Guide) is to be defined as the concentration of the **mixture**, or as the concentration of one of the mixture's constituent species.

Low Pressure Arrhenius Rate

contains inputs for specifying low-pressure Arrhenius parameters.

ln(Pre-exponential Factor)

is the natural logarithm of the constant A_{low} in Equation 7–16 in the Theory Guide. The pre-exponential factor A_{low} is often an extremely large number, so you will input the natural logarithm of this term.

Activation Energy

is the constant E_{low} in Equation 7–16 in the Theory Guide.

Temperature Exponent

is the constant β_{low} in Equation 7–16 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Troe parameters

contains inputs for specifying parameters for the Troe method. See Pressure-Dependent Reactions in the Theory Guide for details. (This section of the dialog box will appear only if you have selected **troe** as the **Reaction Type**.)

Alpha

is the constant α in Equation 7–21 in the Theory Guide.

T1

is the constant T_1 in Equation 7–21 in the Theory Guide.

T2

is the constant T_2 in Equation 7–21 in the Theory Guide.

T3

is the constant T_3 in Equation 7–21 in the Theory Guide.

SRI Parameters

contains inputs for specifying parameters for the SRI method. See Pressure-Dependent Reactions in the Theory Guide for details. (This section of the dialog box will appear only if you have selected **sri** as the **Reaction Type**.)

a

is the constant a in Equation 7–22 in the Theory Guide.

b

is the constant b in Equation 7–22 in the Theory Guide.

c

is the constant c in Equation 7–22 in the Theory Guide.

d

is the constant d in Equation 7–22 in the Theory Guide.

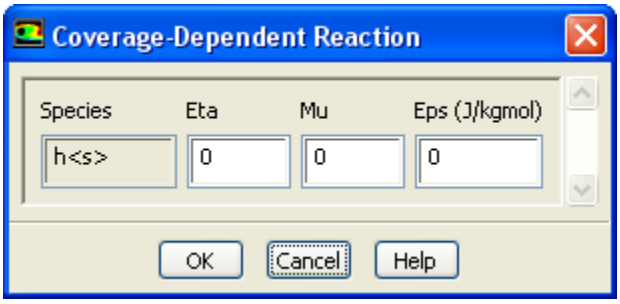
e

is the constant e in Equation 7–22 in the Theory Guide.

35.4.27. Coverage-Dependent Reaction Dialog Box

The **Coverage-Dependent Reaction** dialog box (opened by clicking on **Specify...** under **Coverage-Dependent Reaction** in the *Reactions Dialog Box* (p. 1765)) allows you to model **Wall Surface** reactions with site-balancing.

(Note that the **Coverage-Dependent Reaction** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species
are the site species of the reaction.

Eta
is the surface coverage rate modification of the specie and is defined in Equation 7–45 in the Theory Guide.

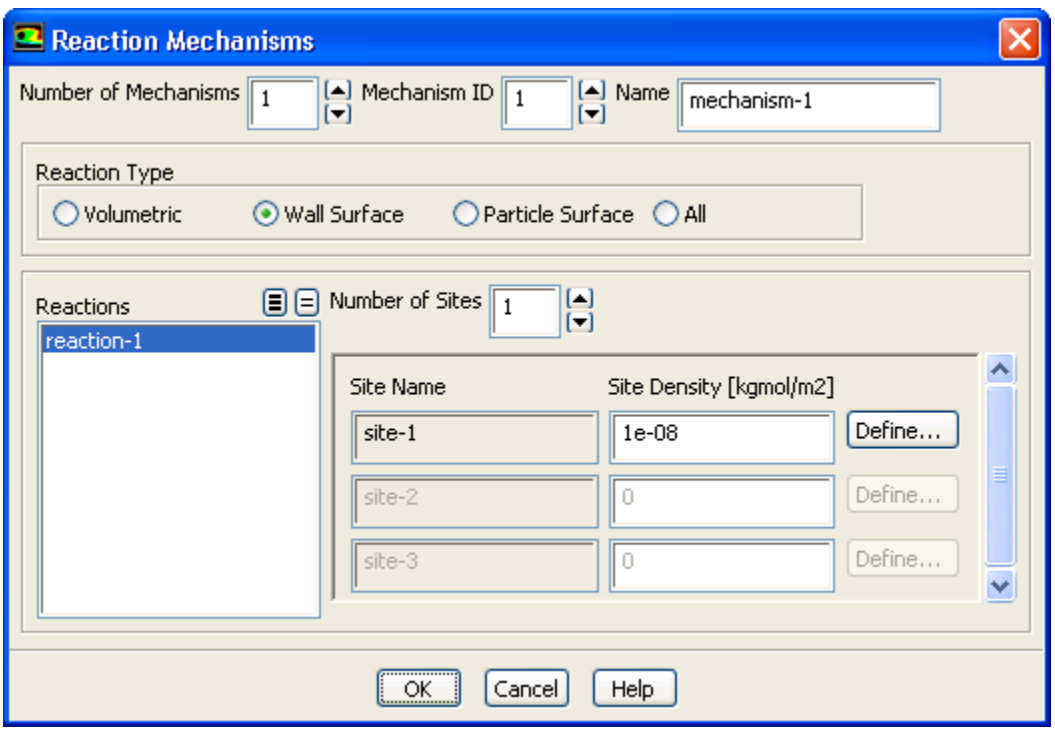
Mu
is the surface coverage rate modification of the specie and is defined in Equation 7–45 in the Theory Guide.

Eps
is the surface coverage rate modification of the specie and is defined in Equation 7–45 in the Theory Guide.

35.4.28. Reaction Mechanisms Dialog Box

The **Reaction Mechanisms** dialog box (opened by clicking on the **Edit...** button next to **Mechanism** in the *Create/Edit Materials Dialog Box* (p. 1736)) allows you to select the reaction mechanism at a particular zone. See *Mixture Materials* (p. 800) for details about these methods and the related inputs.

(Note that the **Reaction Mechanisms** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Number of Mechanisms
specifies the number of mechanisms present.

Mechanism ID
is the ID of the mechanism that you specify.

Name
allows you to enter a name for the mechanism.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction Type
specifies the type of reaction to be displayed for the mechanism.

Volumetric
displays all volumetric reactions under the **Reactions** list.

Wall Surface
displays all wall surface reactions under the **Reactions** list.

Particle Surface
displays all particle surface reactions under the **Reactions** list.

All
displays all types of reactions under the **Reactions** list.

Reactions
displays the list of reactions of the category specified under **Reaction Type**.

Number of Sites
specifies the number of sites at which you can specify the reaction.

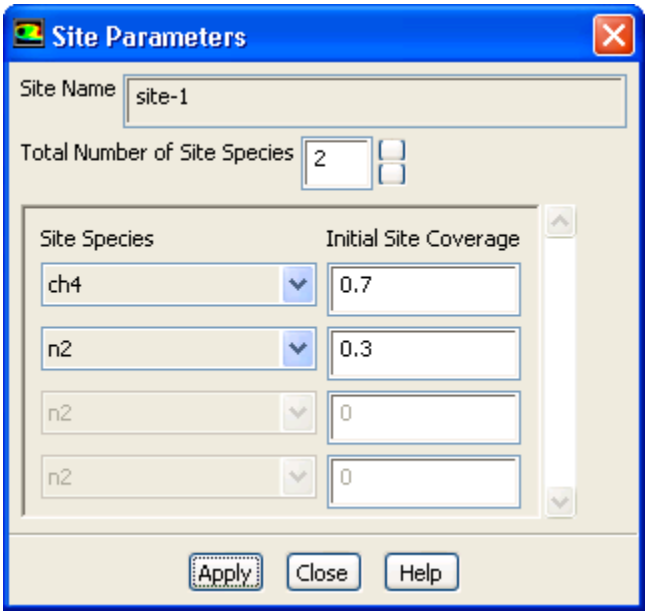
Site Name
contains the name of the site.

Site Density
allows you to specify the site density of the species

Define...
opens the *Site Parameters Dialog Box* (p. 1772).

35.4.29. Site Parameters Dialog Box

The **Site Parameters** dialog box (opened by clicking on the **Define...** button next to **Site Density** in the *Reaction Mechanisms Dialog Box* (p. 1771)) allows you to define the coverage for each site species.



Controls

Site Name
displays the name of site.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total Number of Site Species
specifies the total number of site species.

Site Species
allows you to select the site species.

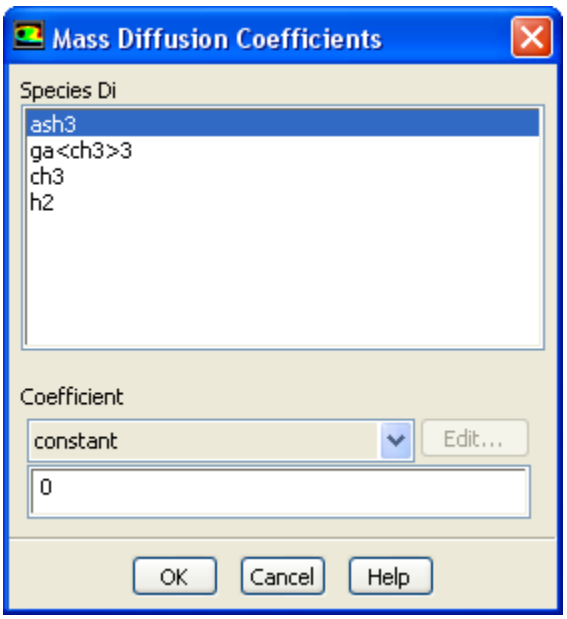
Initial Site Coverage
allows you to specify the initial coverage of the site species.

35.4.30. Mass Diffusion Coefficients Dialog Box

The **Mass Diffusion Coefficients** dialog box (opened by clicking on the **Edit...** button next to **Mass Diffusivity** in the *Create/Edit Materials Dialog Box* (p. 1736)allows you to define the diffusion coefficients of the species in the mixture. Its contents will depend on the method you selected for **Mass Diffusivity**. See *Mass Diffusion Coefficient Inputs* (p. 442) for details about these methods and the related inputs.

(Note that the **Mass Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

For the **dilute-approx** method:



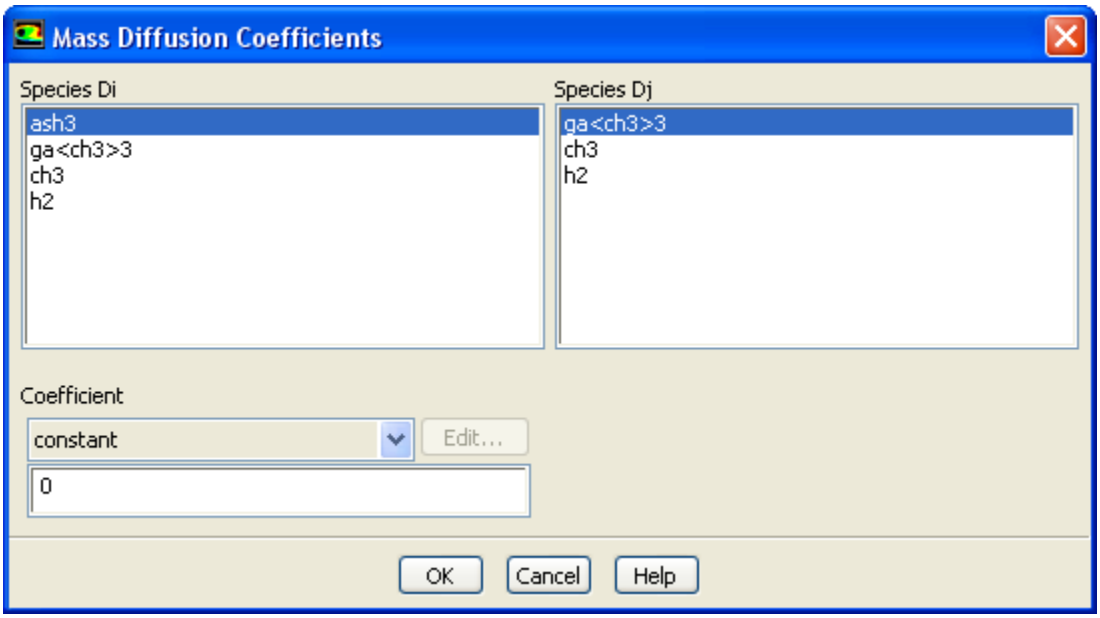
Controls

Species Di
contains a selectable list of all species in the mixture, from which you can select each species and specify its diffusion coefficient.

Coefficient
sets the diffusion coefficient for the selected species in the mixture.

For the **multicomponent** method:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species Di, Species Dj
contain selectable lists of species in the mixture, from which you can select each pair of species and specify the diffusion coefficient of the selected **Species Di** in the selected **Species Dj**.

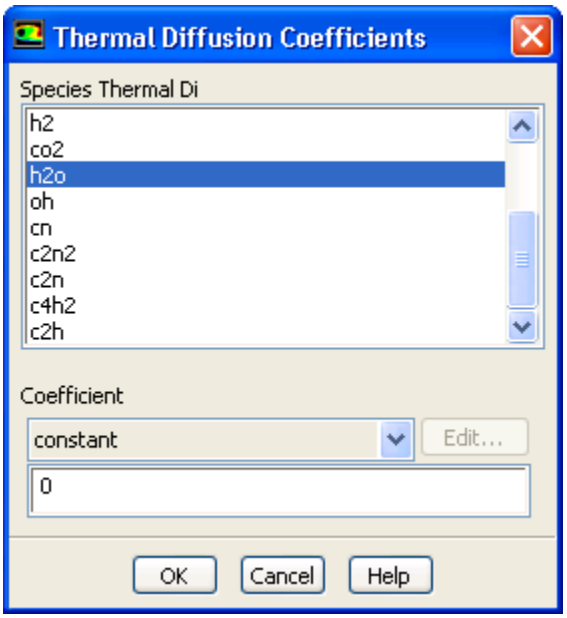
Coefficient
sets the diffusion coefficient for **Species Di** in **Species Dj** (which is equivalent to the diffusion coefficient for **Species Dj** in **Species Di**).

35.4.31. Thermal Diffusion Coefficients Dialog Box

The **Thermal Diffusion Coefficients** dialog box (opened by clicking on the **Edit...** button next to **Thermal Diffusion Coefficient** in the *Create/Edit Materials Dialog Box* (p. 1736)allows you to define the thermal diffusion coefficients of the species in the mixture. See *Thermal Diffusion Coefficient Inputs* (p. 445) for details about these methods and the related inputs.

(Note that the **Thermal Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species Thermal Di

contains a selectable list of all species in the mixture, from which you can select each species and specify its thermal diffusion coefficient.

Coefficient

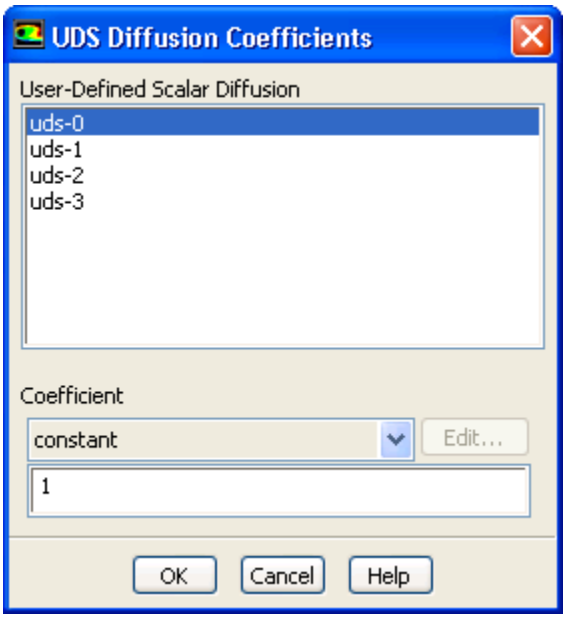
sets the thermal diffusion coefficient for the selected species in the mixture.

35.4.32. UDS Diffusion Coefficients Dialog Box

The **UDS Diffusion Coefficients** dialog box (opened by selecting **uds** and clicking on the **Edit...** button next to **UDS Diffusivity** in the *Create/Edit Materials Dialog Box* (p. 1736)) allows you to define the diffusion coefficients for your user-defined scalar transport equations.

(Note that the **UDS Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



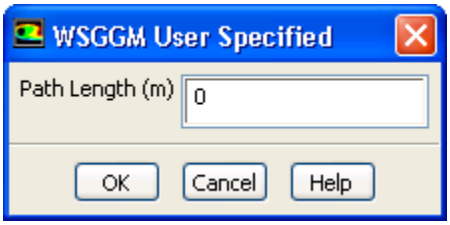
Controls

User-Defined Scalar Diffusion
contains a selectable list of all user-defined scalars, from which you can select each one and specify its diffusion coefficient.

Coefficient
sets the diffusion coefficient for the selected user-defined scalar.

35.4.33. WSGGM User Specified Dialog Box

The **WSGGM User Specified** dialog box allows you to set the path length for the WSGGM when you choose **wsggm-user-specified** as the input method for a composition-dependent **Absorption Coefficient** in the *Create/Edit Materials Dialog Box* (p. 1736). See *Radiation Properties* (p. 435) for details.



Controls

Path Length
sets the path length for the weighted-sum-of-gray-gases model.

35.4.34. Gray-Band Absorption Coefficient Dialog Box

The **Gray-Band Absorption Coefficient** allows you to specify a different absorption coefficient in each gray band when you are modeling non-gray radiation with the P-1 model or the DO model (see The P-1 Model Equations or The DO Model Equations in the Theory Guide and *Setting Up the P-1 Model with Non-Gray Radiation* (p. 706) or *Defining Non-Gray Radiation for the DO Model* (p. 721)). This dialog box will open when you select **gray-band** in the drop-down list next to **Absorption Coefficient** in the *Create/Edit Materials Dialog Box* (p. 1736).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

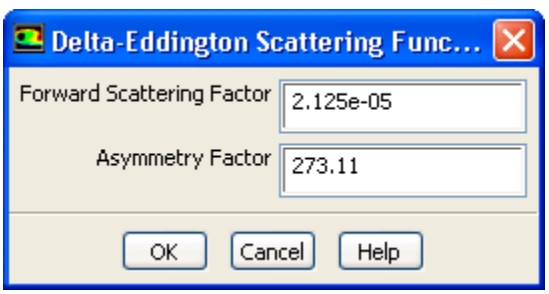
Controls

band n

specifies the absorption coefficient for the nth gray band.

35.4.35. Delta-Eddington Scattering Function Dialog Box

The **Delta-Eddington Scattering Function** dialog box allows you to define the parameters used in the Delta-Eddington phase function for radiation scattering. This dialog box will open when you select **delta-eddington** in the drop-down list next to **Scattering Phase Function** in the *Create/Edit Materials Dialog Box* (p. 1736). See Anisotropic Scattering for details about the items below.



Controls

Forward Scattering Factor

specifies the value of f in Equation 5–68 in the Theory Guide.

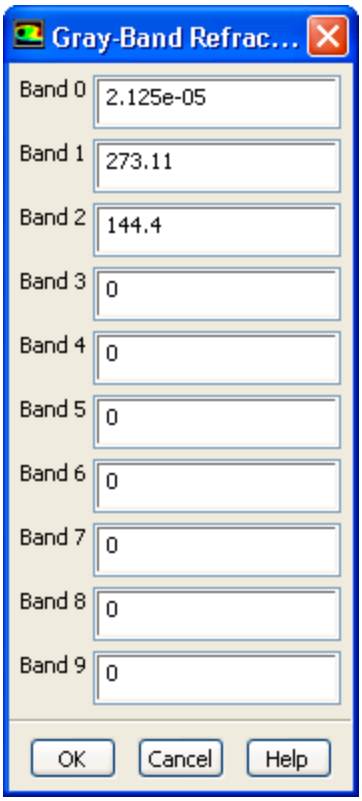
Asymmetry Factor

specifies the value of C in Equation 5–68 in the Theory Guide.

35.4.36. Gray-Band Refractive Index Dialog Box

The **Gray-Band Refractive Index** allows you to specify a different refractive index in each gray band when you are modeling non-gray radiation with the P-1 model or the DO model (see The P-1 Model Equations or The DO Model Equations in the Theory Guide and *Setting Up the P-1 Model with Non-Gray Radiation* (p. 706) or *Defining Non-Gray Radiation for the DO Model* (p. 721)). This dialog box will open when you select **refractive-band** in the drop-down list next to **Refractive Index** in the *Create/Edit Materials Dialog Box* (p. 1736).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

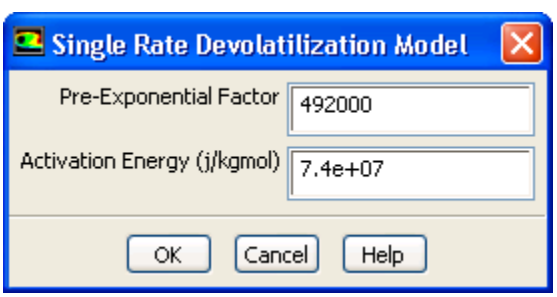
Band n

specifies the refractive index for the nth gray band.

35.4.37. Single Rate Devolatilization Dialog Box

The **Single Rate Devolatilization Model** dialog box (which opens when you select **single-rate** as the **Devolatilization** in the *Create/Edit Materials Dialog Box* (p. 1736) allows you to input the parameters used in the single kinetic rate devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **Single Rate Devolatilization Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Pre-exponential Factor

sets the value of A_1 in Equation 16–99 in the Theory Guide for the computation of the kinetic rate.

Activation Energy

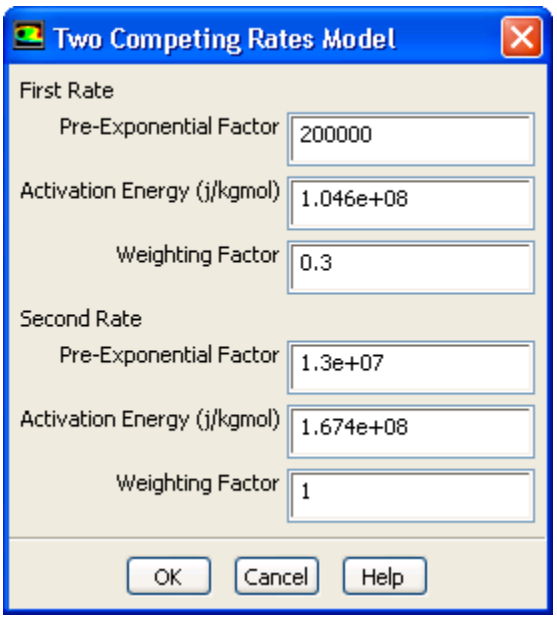
sets the value of E in Equation 16–99 in the Theory Guide for the computation of the kinetic rate.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.38. Two Competing Rates Model Dialog Box

The **Two Competing Rates Model** dialog box (which opens when you select **two-competing-rates** as the **Devolatilization Model** in the *Create/Edit Materials Dialog Box* (p. 1736) allows you to input the parameters used for each of the competing rates in the two-competing-rates devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **Two Competing Rates Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

First Rate

sets parameters for the first of the two rates.

Pre-exponential Factor

sets the value of A_1 in Equation 16–101 in the Theory Guide for the computation of the kinetic rate.

Activation Energy

sets the value of E_1 in Equation 16–101 in the Theory Guide for the computation of the kinetic rate.

Weighting Factor

sets the value of α_1 in Equation 16–103 in the Theory Guide.

Second Rate

sets parameters for the second of the two rates.

Pre-exponential Factor

sets the value of A_2 in Equation 16–102 in the Theory Guide for the computation of the kinetic rate.

Activation Energy

sets the value of E_2 in Equation 16–102 in the Theory Guide for the computation of the kinetic rate.

Weighting Factor

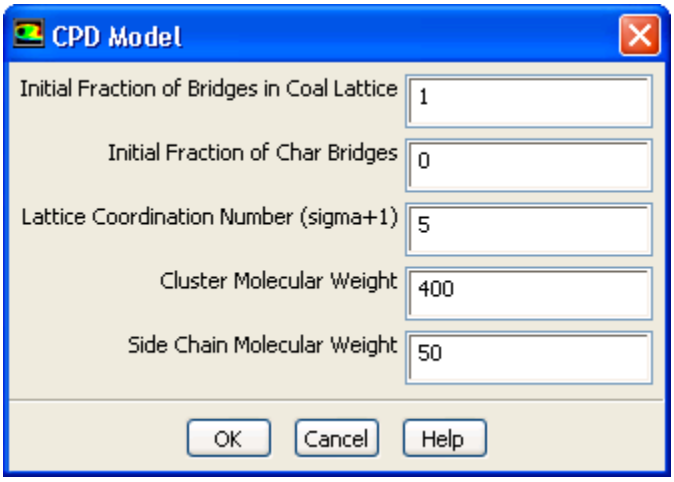
sets the value of α_2 in Equation 16–103 in the Theory Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.39. CPD Model Dialog Box

The **CPD Model** dialog box (which opens when you select **cpd-model** as the **Devolatilization Model** in the *Create/Edit Materials Dialog Box* (p. 1736)) allows you to input the parameters used in the CPD devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **CPD Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Initial Fraction of Bridges in Coal Lattice
sets the value of p_0 in Equation 16–114 in the Theory Guide.

Initial Fraction of Char Bridges
sets the value of c_0 in Equation 16–113 in the Theory Guide.

Lattice Coordination Number
sets the value of $\sigma + 1$ in Equation 16–125 in the Theory Guide.

Cluster Molecular Weight
sets the value of $M_{w,1}$ in Equation 16–125 in the Theory Guide.

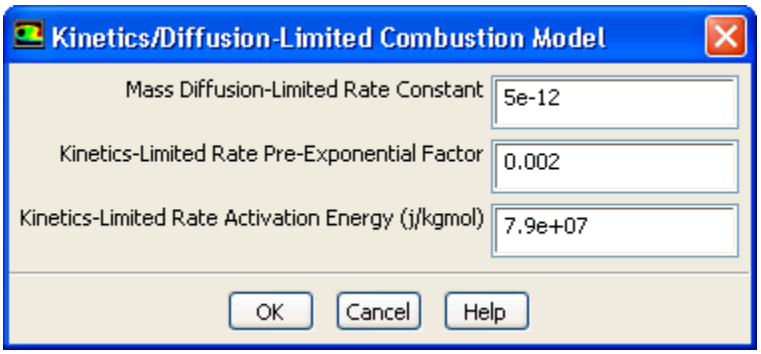
Side Chain Molecular Weight
sets the value of $M_{w,\delta}$ in Equation 16–124 in the Theory Guide.

35.4.40. Kinetics/Diffusion-Limited Combustion Model Dialog Box

The **Kinetics/Diffusion-Limited Combustion Model** dialog box (which opens when you select **kinetics/diffusion-limited** as the **Combustion Model** in the *Create/Edit Materials Dialog Box* (p. 1736) allows you to input the parameters used for the kinetics/diffusion-limited rate surface combustion model. See Surface Combustion (Law 5) in the Theory Guide for details.

Note that the **Kinetics/Diffusion-Limited Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mass Diffusion-Limited Rate Constant

sets the value for C_1 in Equation 16–137 in the Theory Guide.

Kinetics-Limited Rate Pre-exponential Factor

sets the value for C_2 in Equation 16–138 in the Theory Guide.

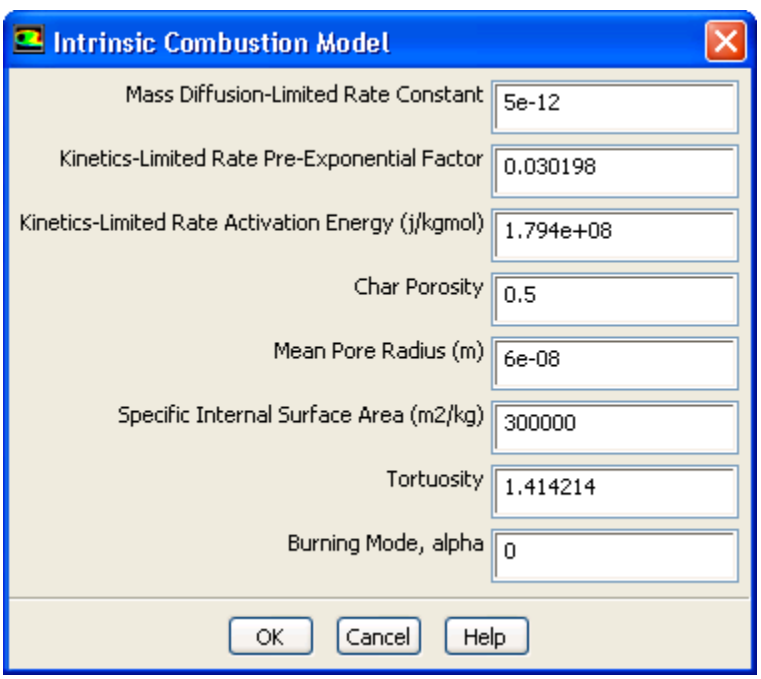
Kinetics-Limited Rate Activation Energy

sets the value for E in Equation 16–138 in the Theory Guide.

35.4.41. Intrinsic Combustion Model Dialog Box

The **Intrinsic Combustion Model** dialog box (which opens when you select **intrinsic-model** as the **Combustion Model** in the *Create/Edit Materials Dialog Box* (p. 1736) allows you to input the parameters used for the intrinsic surface combustion model. See Surface Combustion (Law 5) in the Theory Guide for details.

Note that the **Intrinsic Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass Diffusion-Limited Rate Constant
sets the value for C_1 in Equation 16–137 in the Theory Guide.

Kinetics-Limited Rate Pre-exponential Factor
sets the value for A_i in Equation 16–147 in the Theory Guide.

Kinetics-Limited Rate Activation Energy
sets the value for E_i in Equation 16–147 in the Theory Guide.

Char Porosity
sets the value for θ in Equation 16–144 in the Theory Guide.

Mean Pore Radius
sets the value for $\bar{r}_p$ in Equation 16–146 in the Theory Guide.

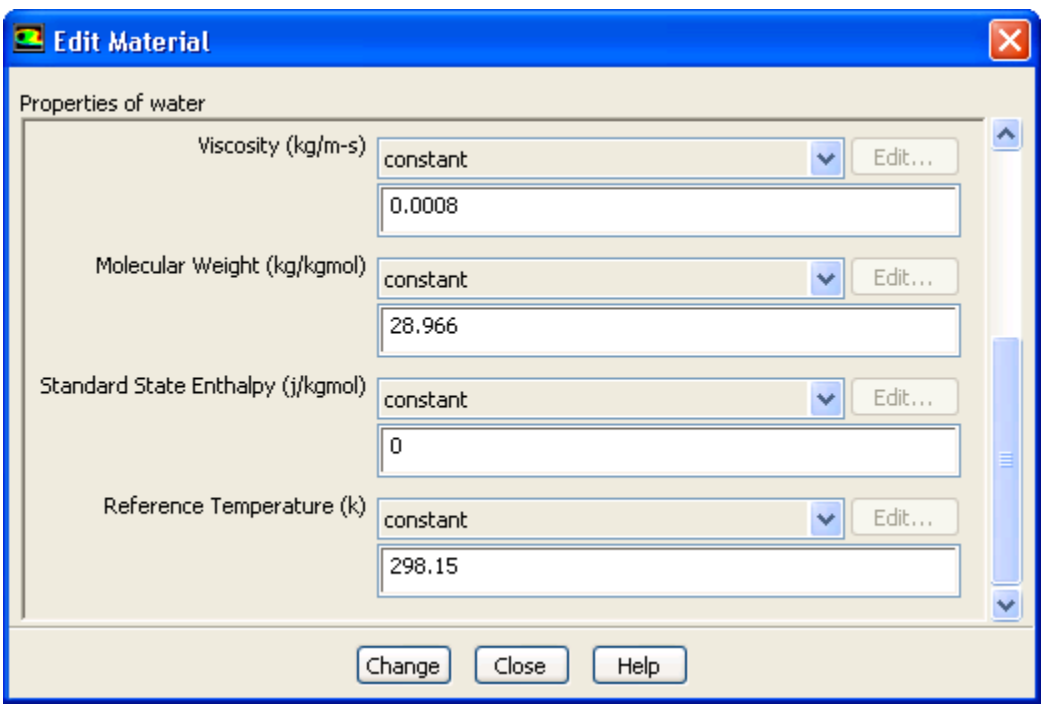
Specific Internal Surface Area
sets the value for A_g in Equation 16–141 and Equation 16–143 in the Theory Guide.

Tortuosity
sets the value for τ in Equation 16–144 in the Theory Guide.

Burning Mode, alpha
sets the value for α in Equation 16–148 in the Theory Guide.

35.4.42. Edit Material Dialog Box

The **Edit Material** dialog box contains the portion of the *Create/Edit Materials Dialog Box* (p. 1736)that contains the properties for a specific material. It is opened from the *Primary Phase Dialog Box* (p. 1784), *Secondary Phase Dialog Box* (p. 1785), *Wall Dialog Box* (p. 1865), *Fluid Dialog Box* (p. 1796), or *Solid Dialog Box* (p. 1802).



Controls

Properties of material-n
contains a list of the properties of material-n. The items in the list are the same as those in the *Create/Edit Materials Dialog Box* (p. 1736).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

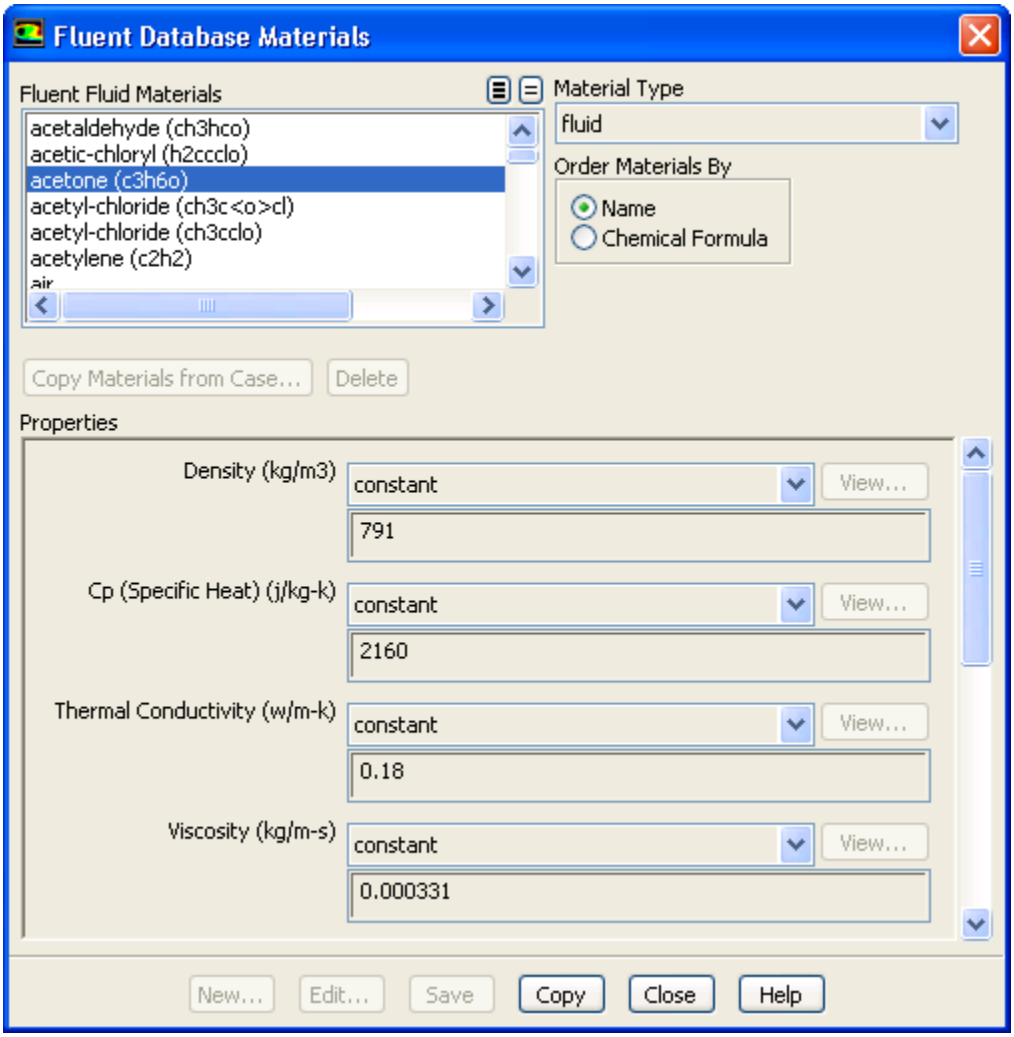
Change

applies any changes you have made to the properties of the material.

35.4.43. Fluent Database Materials Dialog Box

The **Fluent Database Materials** dialog box contains the portion of the *FLUENT Database Materials Dialog Box* (p. 1744) that contains the properties for a specific material. It is opened from the *Species Model Dialog Box* (p. 1681).

*There are two dialog boxes with name **FLUENT Database Materials**. The other dialog box opens up when you click the **FLUENT Database...** button in the Create/Edit Materials Dialog Box (p. 1736).*



Controls

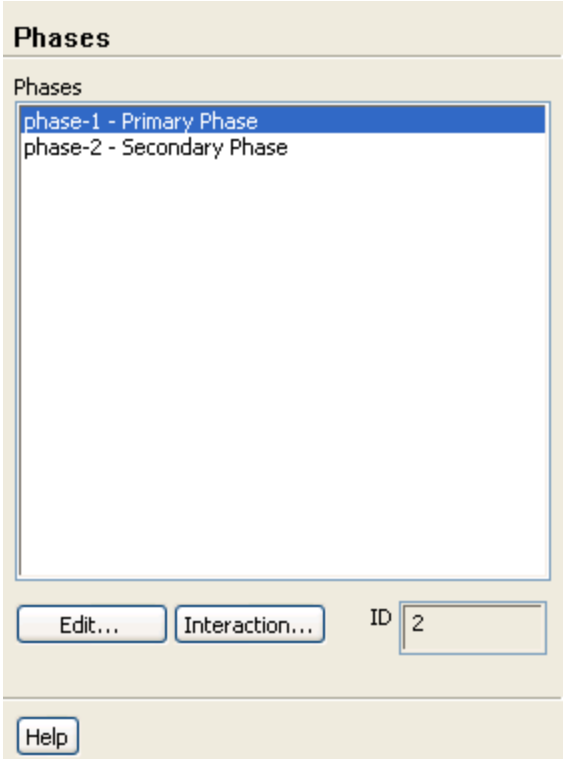
Database Properties of material-n

contains a list of the properties of material-n. The items in the list are the same as those in the *FLUENT Database Materials Dialog Box* (p. 1744).

35.5. Phases Task Page

The **Phases** task page allows you to define each of the phases and the interaction between them. See *Defining the Phases* (p. 1098) – *Defining the Phases for the Eulerian Model* (p. 1154) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Phases
contains a list of all of the phases in the problem from which you can select the phase you want to define or modify. A phase can be a **Primary Phase** or a **Secondary Phase**. You cannot change a phase from primary to secondary, or vice versa. Instead, you can redefine the properties of the primary phase to reflect the new phase designated as primary, and redefine the secondary phases accordingly as well.

Edit...
opens either the *Primary Phase Dialog Box* (p. 1784) or the *Secondary Phase Dialog Box* (p. 1785), where you can define the properties of the selected primary or secondary phase.

Interaction...
opens the *Phase Interaction Dialog Box* (p. 1791), where you can define the interaction between the phases (e.g., surface tension if you are using the VOF model, slip velocity functions if you are using the mixture model, or drag functions if you are using the Eulerian model).

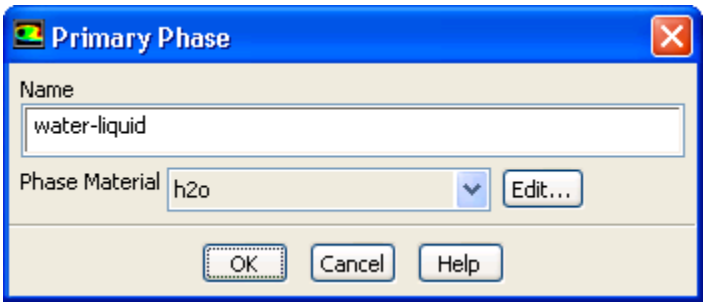
ID
displays the ID number of the phase. You will need this number only if you are writing a user-defined function. See the separate UDF Manual for details about writing user-defined functions for multiphase applications.

For additional information, please see the following sections:
35.5.1. Primary Phase Dialog Box
35.5.2. Secondary Phase Dialog Box
35.5.3. Discrete Phase Dialog Box
35.5.4. Phase Interaction Dialog Box

35.5.1. Primary Phase Dialog Box

The **Primary Phase** dialog box allows you to set the properties of the primary phase. It is opened from the *Phases Task Page* (p. 1783). See *Defining the Phases for the VOF Model* (p. 1136) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
specifies the name of the phase.

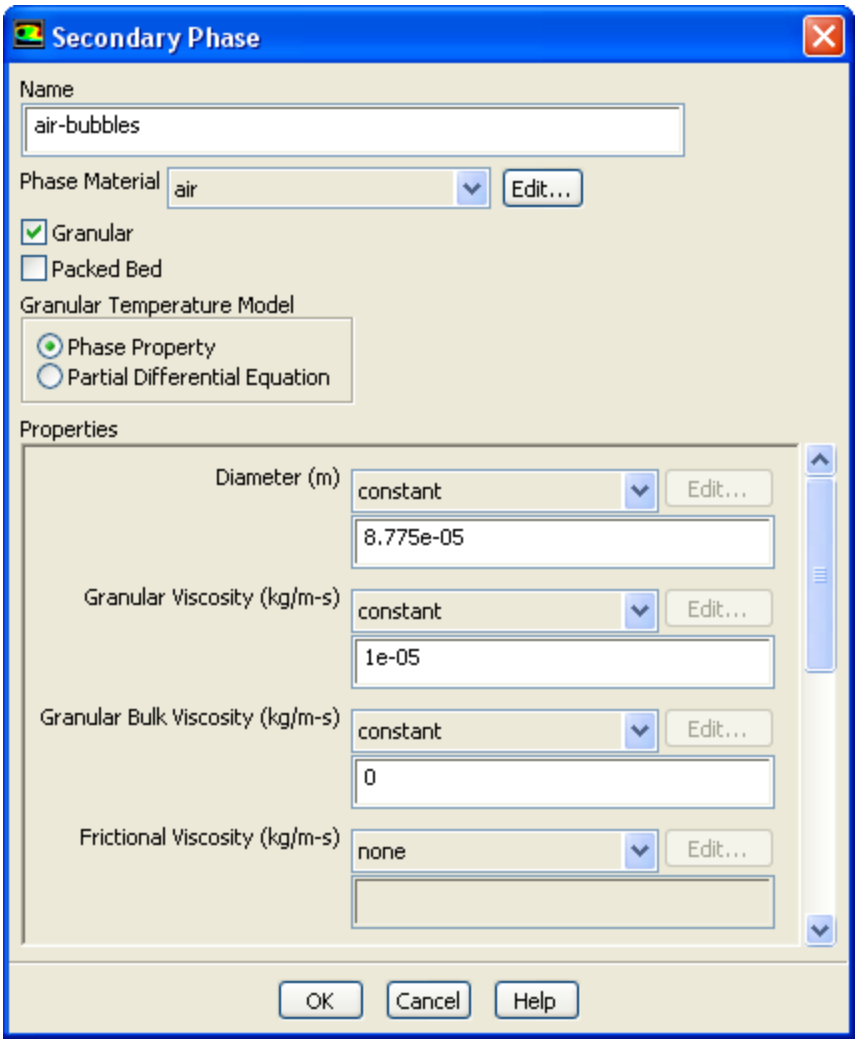
Phase Material
contains a drop-down list of available materials, from which you can select the appropriate one for this phase.

Edit...
opens the *Edit Material Dialog Box* (p. 1782) for the selected **Phase Material**, where you can modify its properties.

35.5.2. Secondary Phase Dialog Box

The **Secondary Phase** dialog box allows you to set the properties of a secondary phase. It is opened from the *Phases Task Page* (p. 1783). The items that appear in the **Secondary Phase** dialog box will depend on which multiphase model you are using. See *Defining the Phases for the VOF Model* (p. 1136), *Defining the Phases for the Mixture Model* (p. 1145), and *Defining the Phases for the Eulerian Model* (p. 1154) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the phase.

Phase Material

contains a drop-down list of available materials, from which you can select the appropriate one for the phase.

Edit...

opens the *Edit Material Dialog Box* (p. 1782) for the selected **Phase Material**, where you can modify its properties.

Granular

indicates whether or not this is a solid phase. This item appears only for the Eulerian model.

Packed Bed

indicates whether or not the granular phase is a packed bed. This option appears only if **Granular** is enabled.

Granular Temperature Model

lists the granular temperature models.

Phase Property

enables phase property model for granular temperature.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Partial Differential Equation

enables partail differential equation model for granular temperature. See Granular Temperature in the Theory Guide for details.

Interfacial Area Concentration

is used to predict mass, momentum and energy transfers through the interface between the phases. See Interfacial Area Concentration in the Theory Guide for details.

Properties

contains a list of phase-specific properties. This section of the dialog box will not appear for the VOF model. The **Diameter** appears for both the mixture model and the Eulerian model, but all of the others will appear only for a granular phase with the Eulerian model.

Diameter

specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the separate UDF Manual for details about user-defined functions.

Granular Viscosity

specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 17–198 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 17–200 in the Theory Guide, select **gidaspow** to compute the value using Equation 17–201 in the Theory Guide, or select **user-defined** to use a user-defined function.

Granular Bulk Viscosity

specifies the solids bulk viscosity (λ_q in Equation 17–121 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 17–202 in the Theory Guide, or select **user-defined** to use a user-defined function.

Frictional Viscosity

specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 17–198 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 17–203 in the Theory Guide, **johnson-et-al**, or select **user-defined** to use a user-defined function.

Angle Of Internal Friction

specifies a constant value for the angle ϕ used in Schaeffer's expression for frictional viscosity (Equation 17–203 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer**, **johnson-et-al**, or **user-defined** for the **Frictional Viscosity**.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17–208 in the Theory Guide, **syamlal-et-al** to apply Equation 17–148 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [*20*] (p. 2210). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the separate UDF Manual for information on hooking a UDF.

Frictional Modulus

can be set as **derived**, or as a **user-defined** function. This is defined as *Equation 26–5* (p. 1148).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5.

Granular Conductivity

specifies the solids conductivity. You can select **syamlal-obrien**, **gidaspow**, **constant** or **user-defined**.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. You can choose either the **algebraic**, the **constant**, or **user-defined** option.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastapour**, or a **user-defined** option.

Elasticity Modulus

is defined as

$$G = \frac{\partial P_s}{\partial \alpha_s} \tag{35–1}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres the packing limit is about 0.63, which is the default value in ANSYS FLUENT. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

Surface Tension

specifies the attractive forces between the interfaces.

Coalescence Kernel

allows you to specify the coalescence kernel. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, or **user-defined**. The two options, **hibiki-ishii** and **ishii-kim**, are described in detail in Interfacial Area Concentration in the Theory Guide.

Breakage Kernel

allows you to specify the breakage kernel. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, or **user-defined**. The two options, **hibiki-ishii** and **ishii-kim**, are described in detail in Interfacial Area Concentration in the Theory Guide.

Dissipation Function

gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose amongst **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

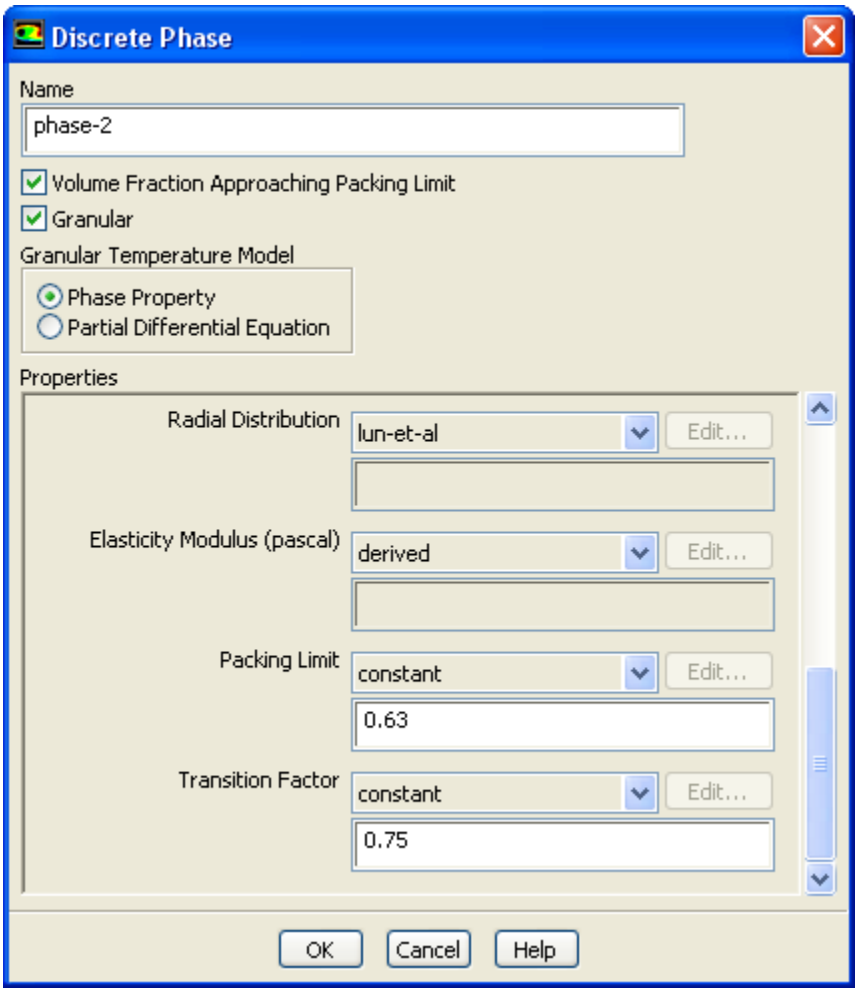
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hydraulic Diameter
is the value used in *Equation 26–7* (p. 1151). This is available when the **wu-ishii-kim** formulation is selected as the **Dissipation Function**.

Min/Max Diameter
are the limits of the bubble diameters.

35.5.3. Discrete Phase Dialog Box

The **Discrete Phase** dialog box allows you to set the properties of a discrete phase. It is opened from the *Phases Task Page* (p. 1783). The items will appear in the **Discrete Phase** dialog box when the **Dense Discrete Phase Model** option is enabled in the **Multiphase Model** dialog box. See *Including the Dense Discrete Phase Model* (p. 1167) for details about the items below.



Controls

Name
specifies the name of the phase.

Volume Fraction Approaching Continuous Flow Limit
appears for non-granular flows. When this option is enabled, only the **Transition Factor** needs to be specified.

Volume Fraction Approaching Packing Limit
prevents the unlimited accumulation of particles, which are operating at packing limit conditions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular
indicates whether or not this is a solid phase. This item appears only for the Eulerian model.

Granular Temperature Model
lists the granular temperature models.

Phase Property
enables phase property model for granular temperature.

Partial Differential Equation
enables partial differential equation model for granular temperature. See Granular Temperature in the Theory Guide for details.

Properties
contains a list of phase-specific properties.

Granular Viscosity
specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 17–198 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 17–200 in the Theory Guide, select **gidaspow** to compute the value using Equation 17–201 in the Theory Guide, or select **user-defined** to use a user-defined function.

Granular Bulk Viscosity
specifies the solids bulk viscosity (λ_q in Equation 17–121 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 17–202 in the Theory Guide, or select **user-defined** to use a user-defined function.

Frictional Viscosity
specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 17–198 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 17–203 in the Theory Guide, **johnson-et-al**, or select **user-defined** to use a user-defined function.

Angle Of Internal Friction
specifies a constant value for the angle ϕ used in Schaeffer's expression for frictional viscosity (Equation 17–203 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer**, **johnson-et-al**, or **user-defined** for the **Frictional Viscosity**.

Frictional Pressure
specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17–208 in the Theory Guide, **syamlal-obrien** to apply Equation 17–148 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [*20*] (p. 2210). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the separate UDF Manual for information on hooking a UDF.

Frictional Modulus
can be set as **derived**, or as a **user-defined** function. This is defined as *Equation 26–5* (p. 1148).

Friction Packing Limit
specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Conductivity

specifies the solids conductivity. You can select **syamlal-obrien**, **gidaspow**,or **user-defined**.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastapour**, or a **user-defined** option.

Elasticity Modulus

is defined as

$$G = \frac{\partial P_s}{\partial \alpha_s} \tag{35–2}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres the packing limit is about 0.63, which is the default value in ANSYS FLUENT. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

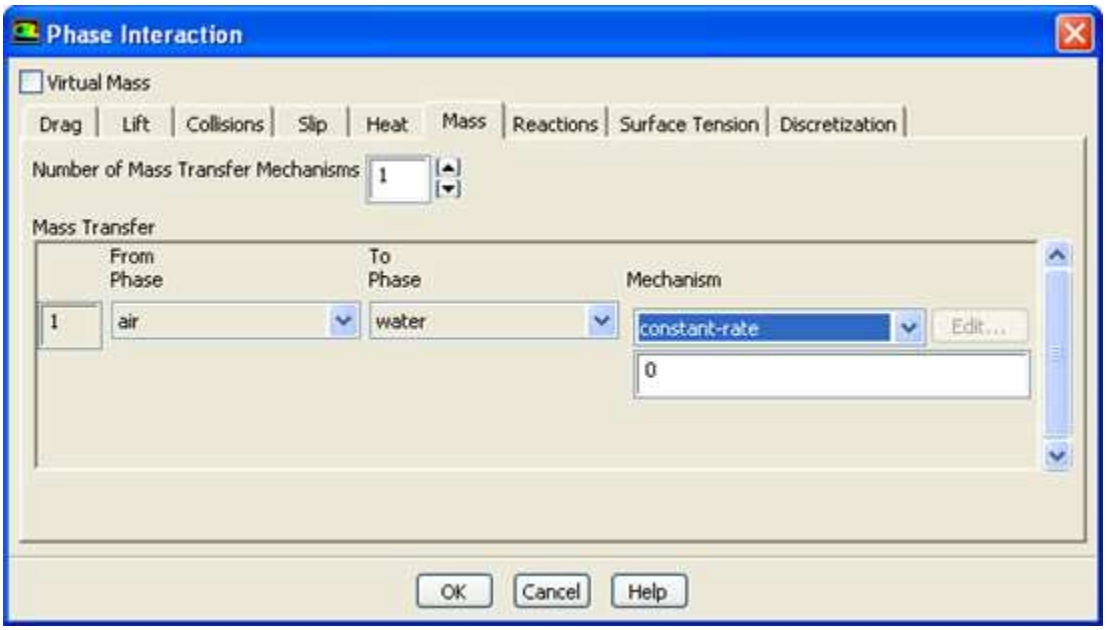
Transition Factor

is specified as either a **constant** or a **user-defined** function. The default value is assumed to be 0.75, which corresponds to the closest sphere packing for monosized spheres (a factor of 4/3). In other words, the transition criterion is based on the local particle volume fraction of the given discrete phase and is specified as a factor multiplied by the maximum packing limit (also a user specified value).

35.5.4. Phase Interaction Dialog Box

The **Phase Interaction** dialog box allows you to define the interaction between phases. It is opened from the *Phases Task Page* (p. 1783). The items that appear in the **Phase Interaction** dialog box will depend on which multiphase model you are using. See *Defining the Phases for the VOF Model* (p. 1136), *Defining the Phases for the Mixture Model* (p. 1145), and *Defining the Phases for the Eulerian Model* (p. 1154) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Virtual Mass

includes the "virtual mass force" (F_{vm} in Equation 17–124 in the Theory Guide) that is present when a secondary phase accelerates relative to the primary phase. This item appears only for the Eulerian model.

Drag

displays the **Drag Coefficient** inputs. This tab is active only for the Eulerian and mixture models.

Drag Coefficient

specifies the drag function for each pair of phases. This section of the dialog box appears only for the Eulerian and mixture models. See *Defining the Phases for the Eulerian Model* (p. 1154) for information about the available options.

Lift

displays the **Lift Coefficient** inputs. This tab is active only for the Eulerian model.

Lift Coefficient

specifies the lift function for each pair of phases. This section of the dialog box appears only for the Eulerian model. See *Defining the Phases for the Eulerian Model* (p. 1154) for information about the available options.

Collisions

displays the **Restitution Coefficient** inputs.

Restitution Coefficient

specifies the restitution coefficient for collisions between each pair of granular phases, and for collisions between particles of the same granular phase. It is relevant only if two or more granular phases are involved. See *Defining the Phases for the Eulerian Model* (p. 1154) for information about the available options.

Slip

displays the **Slip Velocity** inputs. This tab is active only for the mixture model.

Slip Velocity

specifies the slip velocity function for each secondary phase with respect to the primary phase. See *Defining the Phases for the Mixture Model* (p. 1145) for information about the available options.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat

displays the **Heat Transfer Coefficient** inputs. This tab is active only for the Eulerian model when the energy equation is active.

Heat Transfer Coefficient

specifies the heat transfer coefficient function between each pair of phases. See *Including Heat Transfer Effects* (p. 1165) for information about the available options.

Mass

displays the **Mass Transfer Function** inputs.

Number of Mass Transfer Mechanisms

specifies the number of mass transfer mechanisms in your simulation. See *Including Mass Transfer Effects* (p. 1104) for information about the available options.

Reactions

allows you to define multiple heterogeneous reactions and stoichiometry.

Total Number of Heterogeneous Reactions

specifies the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions). See *Specifying Heterogeneous Reactions* (p. 1101) for information about the available options.

Heterogeneous Stiff Chemistry Solver

is used in inter-phase reaction mechanisms containing numerically stiff reactions. This option can improve convergence and is available for transient Eulerian multiphase simulations.

Reaction Name

allows you to enter a name for the reaction.

ID

enables you to set the reaction ID for each reaction.

Number of Reactants

allows you to specify the number of reactants that are involved in the reaction.

Number of Products

allows you to specify the number of products that are involved in the reaction.

Phase

drop-down list allows you to select the phase that is involved in the reaction.

Species

drop-down list allows you to select the species.

Stoich. Coefficient

allows you to set the stoichiometric coefficient.

Reaction Rate Function

allows you to choose rate exponents for an Arrhenius-type reaction, a user-defined function, or a population balance mechanism for the reaction rate.

Surface Tension

includes the effects of surface tension along the fluid-fluid interface.

Wall Adhesion

enables the specification of a wall adhesion angle. (The angle itself, as defined in *Figure 26.25* (p. 1141), will be specified in the *Wall Dialog Box* (p. 1865).) This item will appear only for the VOF and Eulerian models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jump Adhesion

enables the treatment of the contact angle specification at the porous jump boundary. (The angle itself will be specified in the *Porous Jump Dialog Box* (p. 1842).)

Surface Tension Coefficients

specify the surface tension coefficient, σ in Equation 17–22 in the Theory Guide, for each pair of phases. See *Defining the Phases for the VOF Model* (p. 1136) for details. This quantity can be assigned to a new input parameter (indicated by the **New Input Parameter...** option in the drop-down list). Clicking this option displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters. You can also assign this quantity to compatible (based on units) input parameters that are already available in the drop-down list.

Discretization

allows you to use the diffusive and anti-diffusive discretization procedure across the distinct interfaces.

Phase Localized Compressive Scheme

enables the compressive discretization scheme in ANSYS FLUENT, where the degree of diffusion/sharpness is controlled through the value of the slope limiters. This item will appear only for the VOF model and for the Eulerian model with immiscible fluid.

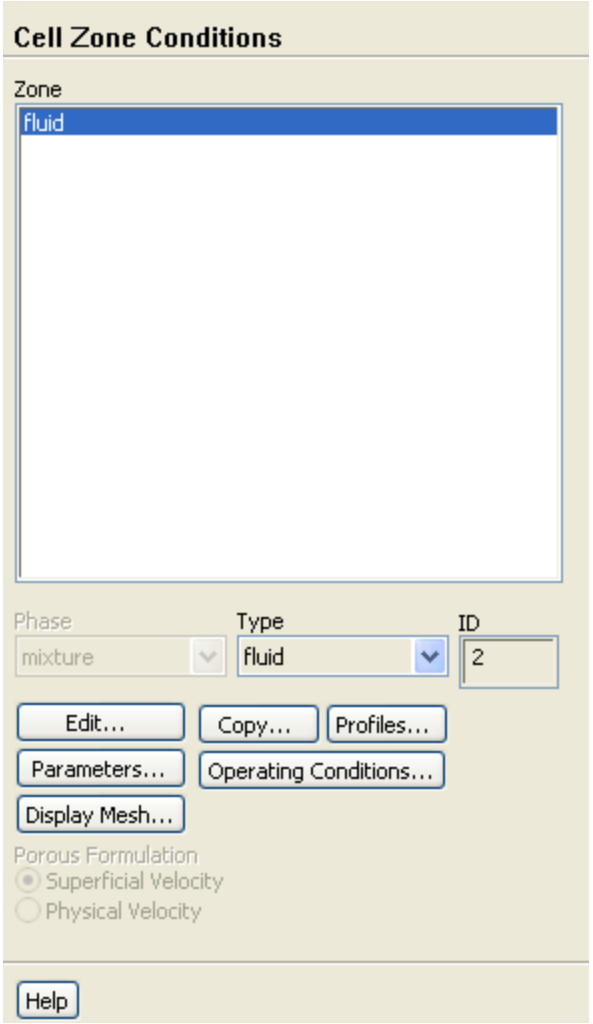
Slope Limiters

are values equating to specific discretization schemes. For each pair of phases, you can enter a value of 0, 1, or 2, or any value between 0 and 2. Depending on the value you use, first order upwind, second order upwind, compressive, or the blended scheme will be applied. Please refer to *Table 26.8: Slope Limiter Discretization Scheme* (p. 1142) to equate each value of the slope limiter with a discretization scheme. For more information, please refer to*Discretizing Using the Phase Localized Compressive Scheme* (p. 1141)

35.6. Cell Zone Conditions Task Page

The **Cell Zones Conditions** task page allows you to set the type of a cell zone and display other dialog boxes to set the cell zone condition parameters for each zone. See *Cell Zone Conditions* (p. 214) for more information.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone

contains a selectable list of available cell zones from which you can select the zone of interest. You can check a zone type by using the mouse probe (see *Controlling the Mouse Button Functions* (p. 1427)) on the displayed physical mesh. This feature is particularly useful if you are setting up a problem for the first time, or if you have two or more cell zones of the same type and you want to determine the cell zone IDs. To do this you must first display the mesh with the *Mesh Display Dialog Box* (p. 1635). Then click the boundary zone with the right (select) mouse button. ANSYS FLUENT will print the cell zone ID and type of that boundary zone in the console window.

Phase

specifies the phase for which conditions at the selected cell **Zone** are being set. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Type

contains a drop-down list of condition types for the selected cell zone. The list contains all possible types to which the cell zone can be changed.

ID

displays the cell zone ID number of the selected cell zone. (This is for informational purposes only; you cannot edit this number.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit...
opens the *Fluid Dialog Box* (p. 1796) or *Solid Dialog Box* (p. 1802).

Copy...
opens the *Copy Conditions Dialog Box* (p. 1805), which allows you to copy conditions from one cell zone to other cell zones of the same type. See *Copying Cell Zone and Boundary Conditions* (p. 208) for details.

Profiles...
opens the *Profiles Dialog Box* (p. 1808).

Parameters...
opens the *Parameters Dialog Box* (p. 2041).

Operating Conditions...
opens the *Operating Conditions Dialog Box* (p. 1806).

Display Mesh...
opens the *Mesh Display Dialog Box* (p. 1635).

Porous Formulation
contains options for setting the velocity in the porous medium simulation. See *Defining the Porous Velocity Formulation* (p. 227) for details.

Superficial Velocity
enables the superficial velocity in a porous medium simulation. This is the default method.

Physical Velocity
enables the physical velocity in a porous medium simulation for a more accurate simulation. This option is available only for a pressure-based solver. See *Modeling Porous Media Based on Physical Velocity* (p. 237) for details.

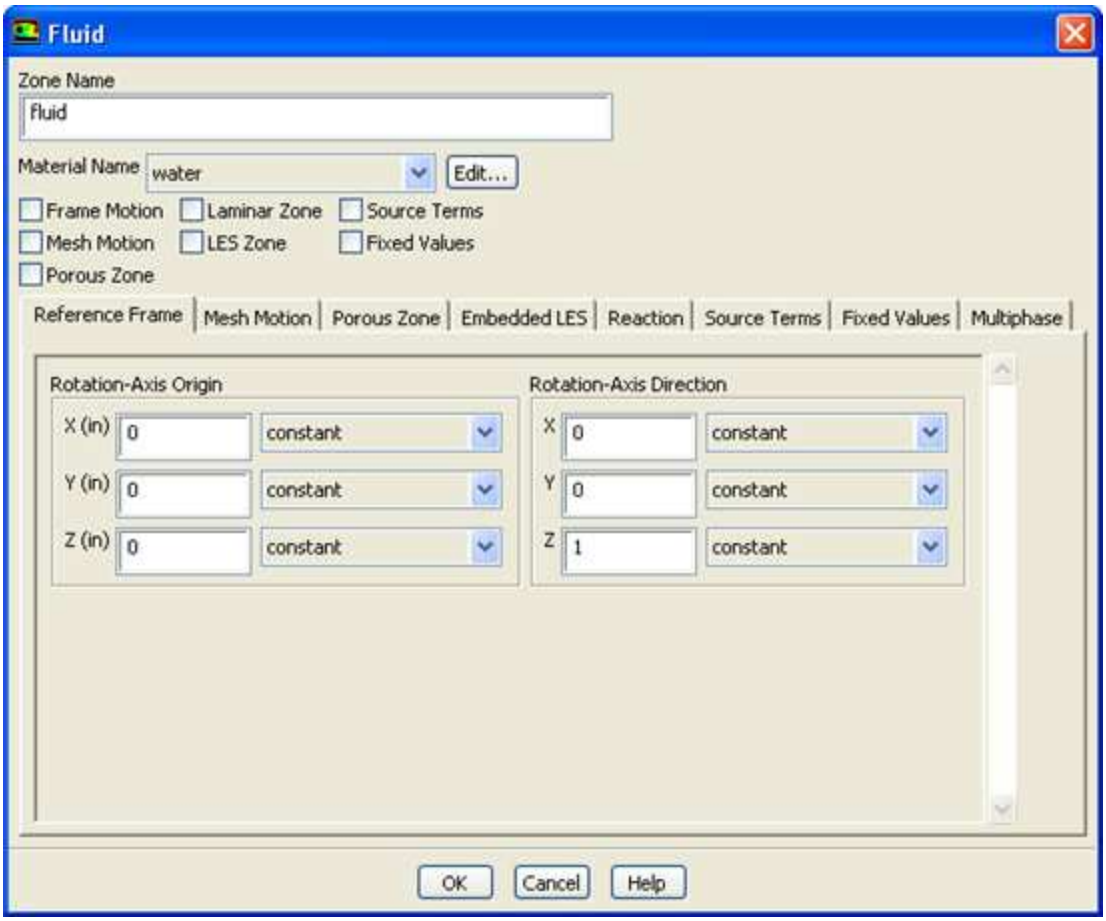
For additional information, please see the following sections:

35.6.1. Fluid Dialog Box
35.6.2. Solid Dialog Box
35.6.3. Copy Conditions Dialog Box
35.6.4. Operating Conditions Dialog Box
35.6.5. Select Input Parameter Dialog Box
35.6.6. Profiles Dialog Box
35.6.7. Orient Profile Dialog Box
35.6.8. Write Profile Dialog Box

35.6.1. Fluid Dialog Box

The **Fluid** dialog box sets the conditions for a fluid cell zone. It is opened from the *Cell Zone Conditions Task Page* (p. 1794). See *Inputs for Fluid Zones* (p. 215) and *User Inputs for Porous Media* (p. 225) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Material Name

sets the fluid material. The drop-down list contains the names of all materials that have been loaded into the solver. Materials are defined with the *Materials Task Page* (p. 1735).

Important

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid zones will be the material you specified in the *Species Model Dialog Box* (p. 1681). For multiphase flows, the materials are specified when you define the phases, as described in *Defining the Phases for the VOF Model* (p. 1136).

Frame Motion

enables the moving reference frame model for the cell zone. See *Specifying the Rotation Axis* (p. 217) and *Defining Zone Motion* (p. 217) for details.

Mesh Motion

enables the sliding mesh model for the cell zone. See *Setting Up the Sliding Mesh Problem* (p. 541) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Porous Zone
indicates that the zone is a porous medium. Additional items will appear in the dialog box when this option is enabled. See *User Inputs for Porous Media* (p. 225) for details.

Laminar Zone
disables the calculation of turbulence production in the fluid zone (appears only for turbulent flow calculations using the Spalart-Allmaras model or one of the k- ε or k- ω models). See *Specifying a Laminar Zone* (p. 216) for details.

LES Zone
allows you to model a smaller embedded LES zone within a larger URANS computational domain for turbulent flow calculations. When you turn on this option, the **Embedded LES** tab will be enabled to allow you to specify properties for the embedded LES zone. See *Setting Up the Embedded Large Eddy Simulation (E-LES) Model* (p. 670) for details.

Source Terms
enables the specification of volumetric sources of mass, momentum, energy, etc. When you turn on this option, the **Source Terms** tab will be enabled to allow you to input the values for the desired sources. See *Defining Mass, Momentum, Energy, and Other Sources* (p. 244) for details.

Fixed Values
enables the fixing of the value of one or more variables in the fluid zone, rather than computing them during the calculation. See *Fixing the Values of Variables* (p. 241) for details.

Important

You can fix values for velocity components, temperature, and species mass fractions only if you are using the pressure-based solver.

Participates In Radiation
specifies whether or not the fluid zone participates in radiation. This option appears when you are using the DO model for radiation.

Reaction
enables/disables reactions in the porous zone.

Reference Frame
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF
allows you to hook the DEFINE_ZONE_MOTION UDF.

Copy to Mesh Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied from one model to the other.

Mesh Motion
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF
allows you to hook the DEFINE_ZONE_MOTION UDF.

Copy to Frame Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied from one model to the other.

Porous Zone
lists the parameters associated with the porous zone.

Conical
enables the specification of a conical (or cylindrical) porous medium. This item will appear only when the **Porous Zone** option is enabled for a 3D case.

Cone Half Angle
specifies the angle between the cone's axis and its surface (see *Figure 7.9* (p. 230)). Set this to 0 to define the porous region using a cylindrical coordinate system. This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Snap to Zone
aligns the plane (or line, in 2D) tool with the zone selected in the drop-down list. The tool is centered at the centroid of the zone, with the tool's axis normal to the zone. If this axis is not the desired cone axis, reposition the tool (as described in *Using the Plane Tool* (p. 1367)). When you are satisfied with the axis, click on the **Update From Plane Tool** (or **Update From Line Tool**) button to update the **Cone Axis Vector** fields.

This item will appear only when the **Porous Zone**and **Conical**options are enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update From Plane Tool

(**Update From Line Tool** in 2D) updates the **Direction-1 Vector** and (in 3D) the **Direction-2 Vector** from the plane tool orientation. If the **Conical** option is enabled, this button will update the **Cone Axis Vector** and the **Point on Cone Axis**. See *Defining the Viscous and Inertial Resistance Coefficients* (p. 228) for details. This item will appear only when the **Porous Zone** option is enabled.

Direction-1 Vector, Direction-2 Vector

indicate the directions for which the resistance coefficients are defined. See *Defining the Viscous and Inertial Resistance Coefficients* (p. 228) for details. These items will appear only when the **Porous Zone** option is enabled, but the **Conical** option is not. (In 2D, only **Direction-1 Vector** will appear.)

Cone Axis Vector

specifies the **X,Y,Z** vector for the cone's axis.

This item will appear only when the **Porous Zone**and **Conical**options are enabled.

Point on Cone Axis

specifies a point on the cone's axis. This point will be used by ANSYS FLUENT to transform the resistances to the Cartesian coordinate system.

This item will appear only when the **Porous Zone**and **Conical**options are enabled.

Relative Velocity Resistance Formulation

allows ANSYS FLUENT to either apply the relative reference frame or the absolute reference frame. This allows for the correct prediction of the source terms.

Viscous Resistance, Inertial Resistance

contain inputs for the viscous resistance coefficient $1/\alpha$ and the inertial resistance coefficient $C2$ in each direction. See *Defining the Viscous and Inertial Resistance Coefficients* (p. 228) for details. These items will appear only when the **Porous Zone** option is enabled.

Alternative Formulation

provides better stability to the calculation when your porous medium is anisotropic.

If you have enabled the **Conical** option, **Direction-1** is the cone axis direction, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) direction.

Power Law Model

contains inputs for the **C0** and **C1** coefficients in the power law model for porous media. See *Using the Power-Law Model* (p. 235) for details.

Fluid Porosity

contains additional inputs for the porous medium. See *User Inputs for Porous Media* (p. 225) for details.

Porosity

sets the volume fraction of fluid within the porous region.

Solid Material Name

specifies the solid material in the porous region.

Reaction

lists the parameters for reactions in the porous zone.

Reaction Mechanism

allows you to specify a defined group, or mechanism, of available reactions. See *Defining Zone-Based Reaction Mechanisms* (p. 816) for details about defining reaction mechanisms.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface-to-Volume Ratio

specifies the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, ANSYS FLUENT can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell.

Source Terms

defines a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone.

Fixed Values

lists the fixed parameters of the fluid zone.

Local Coordinate System for Fixed Velocities

enables the specification of fixed cylindrical velocity components instead of Cartesian components. The local coordinate system is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction**.

This item is available only in 3D, and only when the **Fixed Values** option is on.

Multiphase

allows you to set parameters that are specific to the multiphase models.

Compressive Scheme Slope Limiter

ranges from 0 to 2. For example, a **Compressive Scheme Slope Limiter** of 0 corresponds to first order upwind behavior, a value of 1 corresponds to second order upwind, and a value of 2 applies the compressive scheme. The **Multiphase** tab is available only if **Zonal Discretization** is enabled in the *Multiphase Model Dialog Box* (p. 1642)..

Numeric Beach Treatment

is available when **Open Channel Flow** and/or **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box (see *Numerical Beach Treatment for Open Channels* (p. 1133) for more detail).

Numerical Beach

when enabled expands the dialog box where you can specify the numerical beach parameters.

Beach Group ID

represents the cell zones sharing the damping length containing the same input parameters.

Damping Type

allows you to choose between **One Dimensional** and **Two Dimensional** damping.

One Dimensional

is the damping treatment in the flow direction.

Two Dimensional

is the damping treatment in the flow and gravity direction.

Compute From Inlet Boundary

is set to **none** by default. If there are available open channel boundaries (velocity-inlet, pressure-inlet, and mass-flow-inlet), boundary names are added to the drop-down list. If you select a boundary from the list, the **Level Inputs** and the **Damping Length Inputs in Flow Direction** will be updated in the interface. You have the option to overwrite the updated inputs with values that are more applicable to your simulation.

Level Inputs

is only available for the **Two Dimensional** damping type.

Free Surface Level

is the same definition as for open channel flow, see *Modeling Open Channel Flows* (p. 1121).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bottom Level
is the same definition as for open channel flow, see *Modeling Open Channel Flows* (p. 1121), and is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Flow Direction
is the X, Y, and Z (for 3D) components.

Damping Length Inputs in Flow Direction
are required to calculate the start and end points of the damping length in the flow direction.

Damping Length Specification
is only available if **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box. There are two options you can choose from **End Point and Wave Lengths** or **End and Start Points**.

End Point
is the end point of the damping zone.

Start Point
is the starting point in the flow direction.

Wave Length
is updated automatically if the boundary is selected from the **Compute From Inlet Boundary** drop-down list.

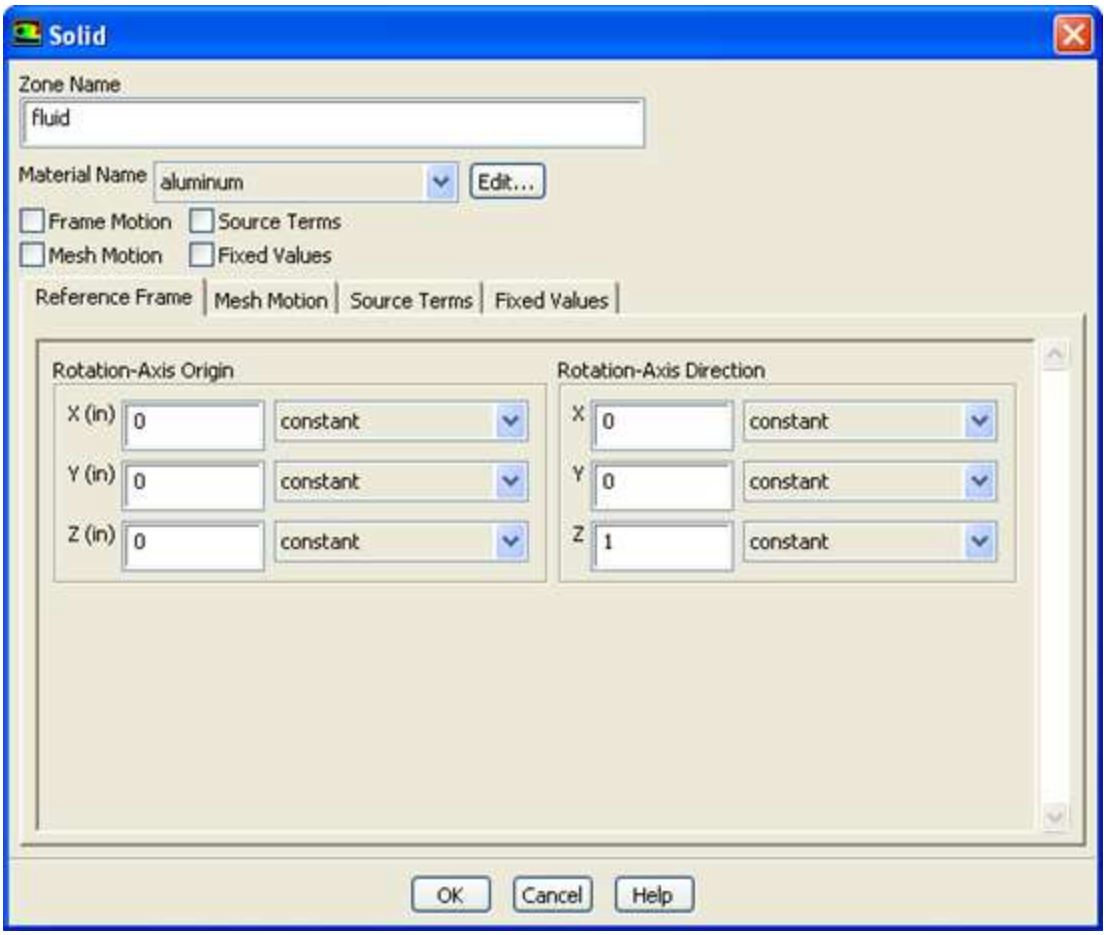
Number Of Wave Lengths
is set to 2 by default for the calculation of the damping length.

Damping Resistance
is the resistance per unit length.

35.6.2. Solid Dialog Box

The **Solid** dialog box sets the boundary conditions for a solid cell zone. It is opened from the *Cell Zone Conditions Task Page* (p. 1794). See *Inputs for Solid Zones* (p. 218) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Material Name

selects the material type of the solid. Materials are defined with the *Materials Task Page* (p. 1735).

Frame Motion

enables the moving reference frame model for the cell zone. See *Specifying the Rotation Axis* (p. 217) and *Defining Zone Motion* (p. 217) for details.

Mesh Motion

enables the sliding mesh model for the cell zone. See *Setting Up the Sliding Mesh Problem* (p. 541) for details.

Source Terms

enables the specification of a volumetric source of energy. When you turn on this option, the **Source Term** tab will allow you to input the value for the energy source. See *Defining Mass, Momentum, Energy, and Other Sources* (p. 244) for details.

Fixed Values

enables the fixing of the value of temperature in the solid zone, rather than computing it during the calculation. See *Fixing the Values of Variables* (p. 241) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

You can fix the value of temperature only if you are using the pressure-based solver.

Participates In Radiation
specifies whether or not the solid zone participates in radiation. This option appears when you are using the DO model for radiation.

Reference Frame
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF
allows you to hook the DEFINE_ZONE_MOTION UDF.

Copy to Mesh Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied from one model to the other.

Mesh Motion
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See *Specifying the Rotation Axis* (p. 219) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF
allows you to hook the DEFINE_ZONE_MOTION UDF.

Copy to Frame Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied from one model to the other.

Source Term
lists the parameters for volumetric source of energy.

Energy
displays the total number of energy sources used.

User Scalar n
displays the total number of scalars used.

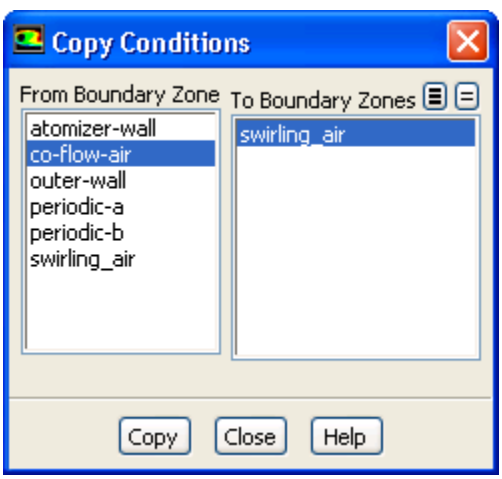
Fixed Values
lists the parameters that can be declared as fixed during the calculation.

Temperature
specifies the fixed value for temperature.

User Scalar n
specifies the fixed value for user scalar.

35.6.3. Copy Conditions Dialog Box

The **Copy Conditions** dialog box allows you to copy cell zone and/or boundary conditions from one zone/boundary to other zones/boundaries of the same type. It is opened either from the *Cell Zone Conditions Task Page* (p. 1794) or from the *Boundary Conditions Task Page* (p. 1812). See *Copying Cell Zone and Boundary Conditions* (p. 208) for details.



Controls

From Zone
specifies the zone that has the conditions you want to copy.

To Zones
specifies the zone or zones to which you want to copy the conditions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase

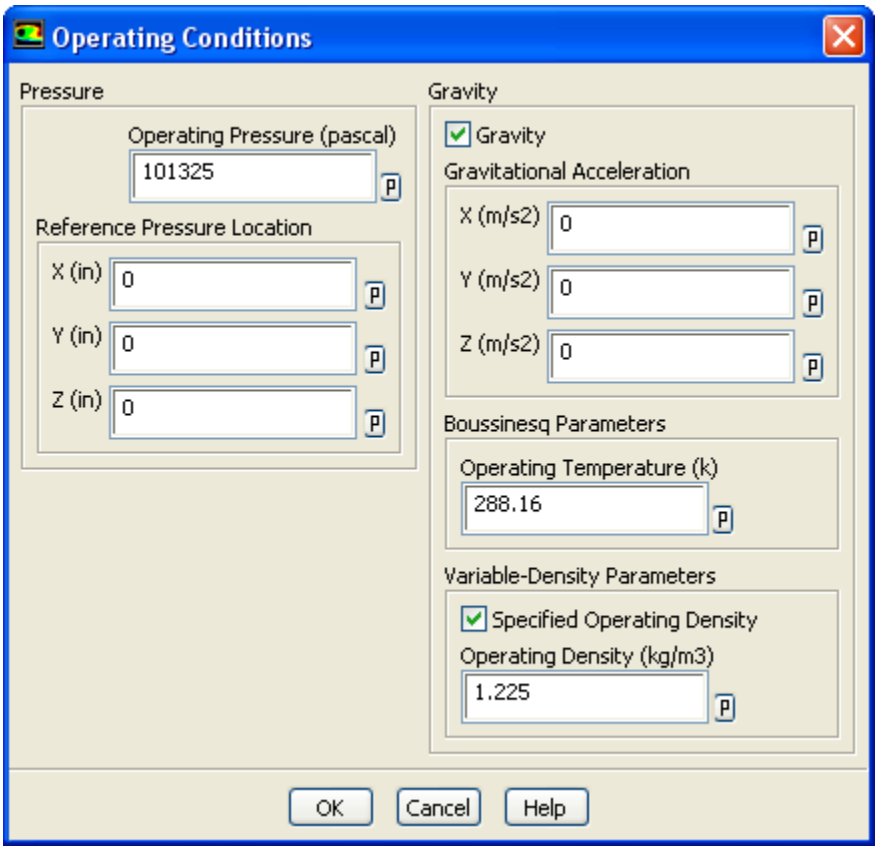
specifies the phase for which cell zone conditions or boundary conditions are being copied. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See *Steps for Copying Cell Zone and Boundary Conditions* (p. 1119) for details.

Copy

copies the cell zone conditions or boundary conditions, setting all of the conditions for the zones selected in the **To Zones** list to be the same as the conditions for the zone selected in the **From Zone** list.

35.6.4. Operating Conditions Dialog Box

The **Operating Conditions** dialog box allows you to set parameters related to operating conditions in your model.



Controls

Pressure

contains items related to the modeling of pressure.

Floating Operating Pressure

specifies the use of a floating operating pressure. See *Floating Operating Pressure* (p. 507) for details. This option appears only for time-dependent compressible flows.

Operating Pressure

sets the operating pressure for the problem. For all flows, ANSYS FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the relative pressure. See *Operating Pressure* (p. 449) for a detailed description of operating pressure and how to set it.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Pressure Location
sets the location of the cell whose pressure value is used to adjust the gauge pressure field for incompressible flows that do not involve any pressure boundaries. See *Reference Pressure Location* (p. 450) for details.

Gravity
contains inputs for gravitational acceleration, the Boussinesq model, and variable density.

Gravity
enables the specification of gravity.

Gravitational Acceleration
sets the x, y, and z components of the gravitational acceleration vector. (The z component is available only in 3D solvers.) See *Natural Convection and Buoyancy-Driven Flows* (p. 697) for details about buoyancy-driven flows. This option appears only when **Gravity** is enabled.

Boussinesq Parameters
contains inputs related to the Boussinesq model. This option appears only if **Energy** (in the *Energy Dialog Box* (p. 1645)) and **Gravity** are enabled. See *The Boussinesq Model* (p. 697) for more information on the Boussinesq model.

Operating Temperature
sets the operating temperature (T_0 in *Equation 14–2* (p. 697)) for use with the Boussinesq approximation.

Variable-Density Parameters
contains inputs related to the modeling of variable density. This option appears only when **Gravity** is enabled.

Specified Operating Density
enables the specification of operating density. See *Operating Density* (p. 700) for details.

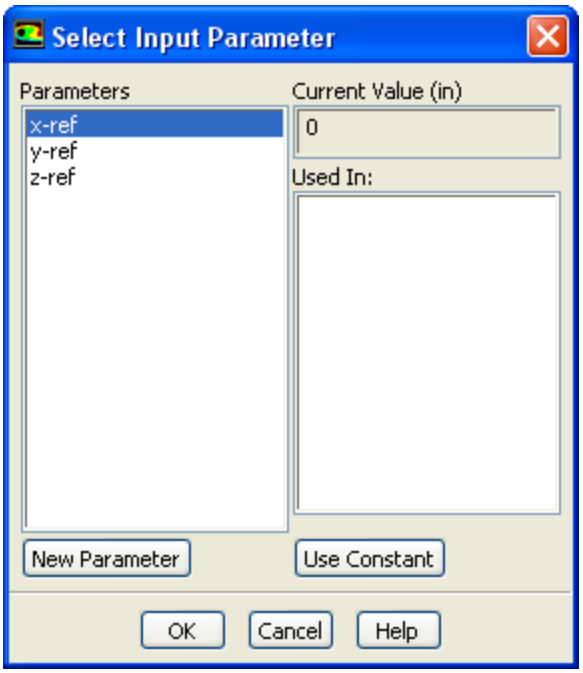
Operating Density
sets the operating density (ρ_0 in *Equation 14–3* (p. 700)). This parameter can be set only when **Specified Operating Density** is enabled.

These quantities can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

35.6.5. Select Input Parameter Dialog Box

The **Select Input Parameter** dialog box allows you to choose from a listing of existing input parameters as well as to create and define new input parameters. For more information about parameters, see *Defining and Viewing Parameters* (p. 210) in the User's Guide, and see Working With Input and Output Parameters in Workbench in the ANSYS FLUENT in Workbench User's Guide.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Parameters
contains a list of existing compatible input parameters

Current Value
displays the value of the currently selected parameter.

Used In
lists any variables that are already associated with the currently selected parameter.

Use Constant
allows you to change the associated parameter to a constant (i.e., real) value.

New Parameter
opens the *Input Parameter Properties Dialog Box* (p. 2043), in which you can assign names and values to an input parameter.

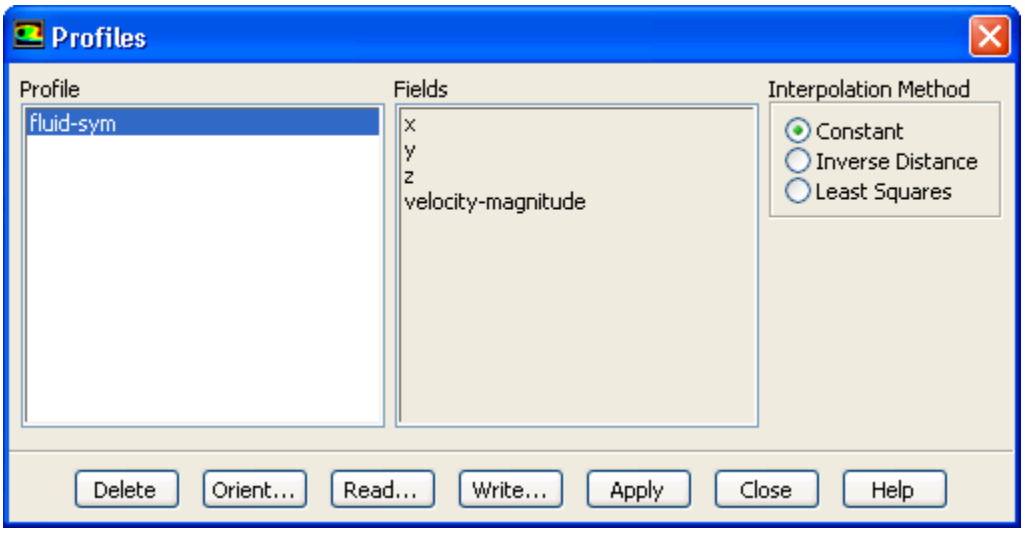
Note

Source term parameters can only be specified using SI units. The default parameter name will not include the unit quantity name. Even if you have selected units other than SI for the source term parameters, you must still provide values assuming SI units for the (source-term) parameters in *both* FLUENT and Workbench.

35.6.6. Profiles Dialog Box

The **Profiles** dialog box allows you to define new profiles by reading cell zone and boundary profile files. You can also examine the existing profile definitions and delete unused profiles. See *Profiles* (p. 366) for details about cell zone and boundary profiles.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Profile

contains a selectable list of available profiles. When a profile is selected its available fields are displayed under **Fields**.

Fields

displays the fields available in the selected profile. After the profile file has been read, these fields will also appear in any boundary condition dialog box (e.g., the **Velocity Inlet** dialog box) that allows profile specification of a variable. To the right of (or below) the variable in the boundary conditions dialog box, there will be a drop-down list that contains a **constant** and the fields from available profile files. To use a particular profile field, just select it from the list.

Interpolation Method

allows you to select the interpolation method for the profile selected from the **Profile** list. This selection is only available for point profiles, and will only take effect when the **Apply** button is clicked. The choices include the following:

Constant

specifies that ANSYS FLUENT should use zeroth-order interpolation to assign the point profile values to the nearest cell faces at the boundary. This is the default selection.

Inverse Distance

specifies that ANSYS FLUENT should assign a value to each cell face at the boundary based on weighted contributions from the values in the point profile file. The weighting factor is inversely proportional to the distance between the profile point and the cell face center.

Least Squares

specifies that ANSYS FLUENT should assign values to the cell faces at the boundary through a first-order interpolation method that tries to minimizes the sum of the squares of the offsets (residuals) between the profile data points and the cell face centers.

Delete

deletes the selected profile from memory.

Orient...

opens the *Orient Profile Dialog Box* (p. 1810), in which you can reorient and scale the profile. This item appears only in 3D.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read...

opens *The **Select File** Dialog Box* (p. 33) so that you can read a boundary profile file. If a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Write...

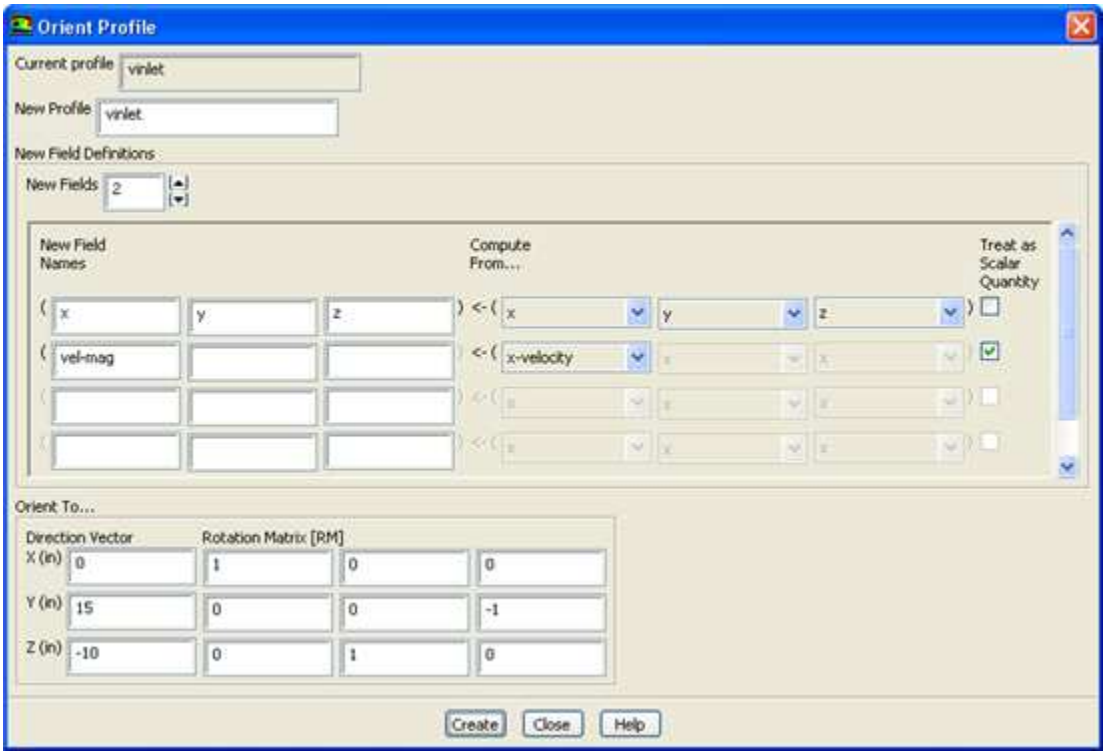
opens the *Write Profile Dialog Box* (p. 1811), in which you can save profile data.

Apply

sets the selection made in the **Interpolation Method** list for point profiles in preparation for interpolation. The profile is not actually interpolated until a profile field is selected in a boundary condition dialog box (e.g., the **Velocity Inlet** dialog box) and the solution is initialized.

35.6.7. Orient Profile Dialog Box

The **Orient Profile** dialog box allows you to reorient a profile so that you can apply it to a particular boundary. See *Reorienting Profiles* (p. 371) for details.



Controls

Current profile

shows the name of the currently selected profile in the *Profiles Dialog Box* (p. 1808). This is the profile on which the new profile will be based.

New Profile

sets the name of the new cell zone or boundary profile.

New Field Definitions

contains inputs and controls for the definition of the vector and scalar fields in the new profile.

New Fields

sets the number of data fields in the new profile.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Field Names

contains inputs for the names of the data fields in the new profile. For a vector field, all 3 inputs in each row will be active; for a scalar field, only the first will be active.

Compute From...

contains drop-down lists with the names of the fields in the **Current profile**. In these lists, select the fields from which the **New Field Names** will be computed.

Treat as Scalar Quantity

indicates (if on) that the adjacent field is a scalar quantity. If this option is off, the field is a vector quantity.

Orient To...

contains inputs for the definition of the local coordinate system for the new profile. This coordinate system will determine the orientation of the profile.

Direction Vector

is the vector that translates a cell zone or boundary profile to the new position, and is defined between the centers of the profile fields.

Rotation Matrix [RM

] specifies the rotational matrix RM which is based on Euler angles (γ, β, and α) that define an orthogonal system $x'y'z'$ as the result of the three successive rotations from the original system xyz. See *Reorienting Profiles* (p. 371) for more information.

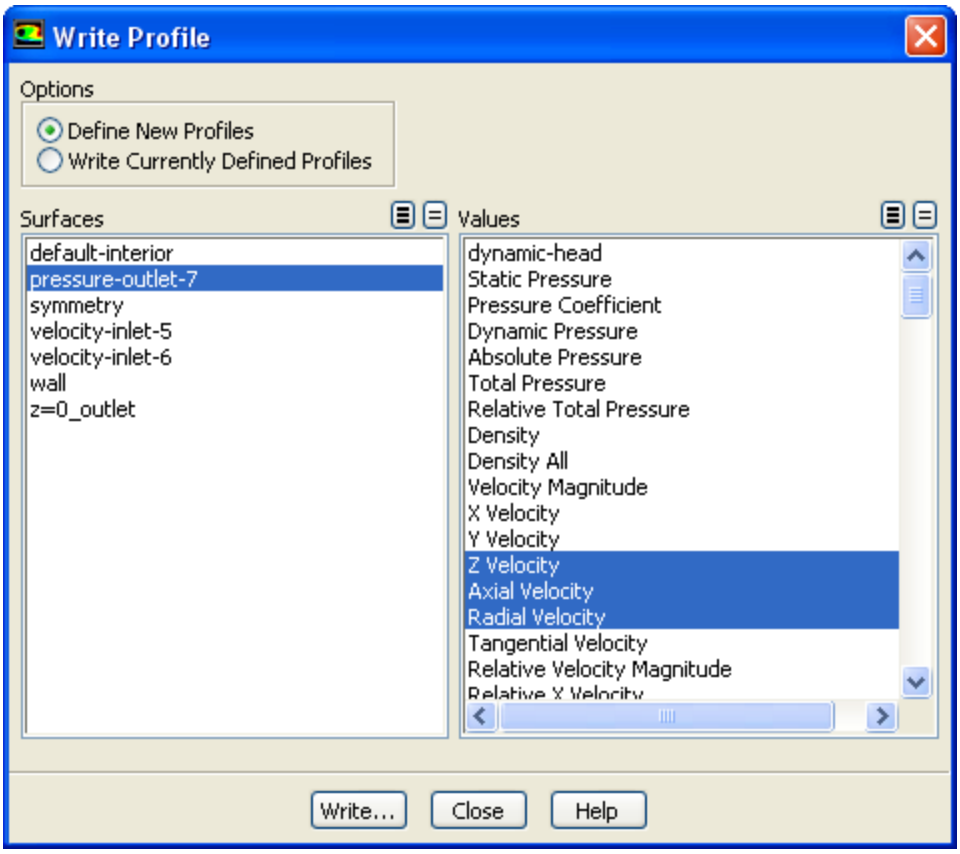
Create

creates a new profile using the information specified in the dialog box.

35.6.8. Write Profile Dialog Box

The **Write Profile** dialog box allows you to create a profile file from the conditions on a specified cell zone or boundary/surface. See *Writing Profile Files* (p. 71) for details on writing profile files.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains options for writing profiles.

Define New Profiles

enables the creation of a profile file from the conditions on a specified boundary or surface.

Write Currently Defined Profiles

enables the creation of a profile file containing all profiles that are currently defined.

Surfaces

contains a list from which you can select the surface(s) from which you want to extract data.

Values

contains a list from which you can select the variable(s) for which you want to create profiles.

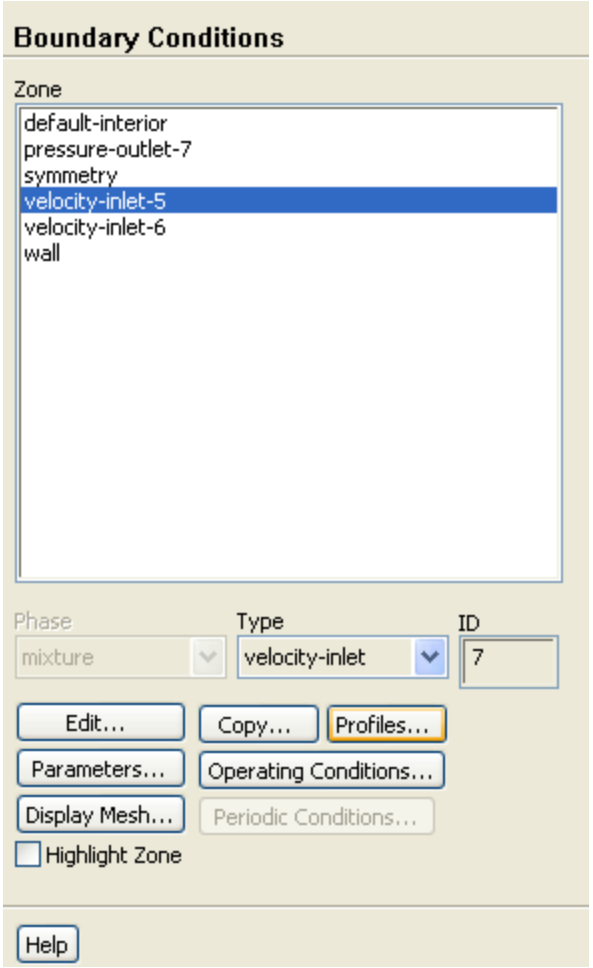
Write...

opens *The **Select File** Dialog Box* (p. 33), in which you can specify a filename for the profile file.

35.7. Boundary Conditions Task Page

The **Boundary Conditions** task page allows you to set the type of a boundary and display other dialog boxes to set the boundary condition parameters for each boundary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone

contains a selectable list of boundary zones from which you can select the zone of interest. You can check a zone type by using the mouse probe (see *Controlling the Mouse Button Functions* (p. 1427)) on the displayed physical mesh. This feature is particularly handy if you are setting up a problem for the first time, or if you have two or more boundary zones of the same type and you want to determine the zone IDs. To do this you must first display the mesh with the *Mesh Display Dialog Box* (p. 1635). Then click the boundary zone with the right (select) mouse button. ANSYS FLUENT will print the zone ID and type of that boundary zone in the console window.

Phase

specifies the phase for which conditions at the selected boundary **Zone** are being set. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Type

contains a drop-down list of boundary condition types for the selected zone. The list contains all possible types to which the zone can be changed.

Important

Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. *Creating Conformal Periodic Zones* (p. 191) explains how to create and uncouple periodic zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ID
displays the zone ID number of the selected zone. (This is for informational purposes only; you cannot edit this number.)

Edit...
opens the appropriate dialog box for setting the boundary conditions for that particular boundary type.

Copy...
opens the *Copy Conditions Dialog Box* (p. 1805), which allows you to copy boundary conditions from one zone to other zones of the same type. See *Copying Cell Zone and Boundary Conditions* (p. 208) for details.

Profiles...
opens the *Profiles Dialog Box* (p. 1808).

Parameters...
opens the *Parameters Dialog Box* (p. 2041).

Operating Conditions...
opens the *Operating Conditions Dialog Box* (p. 1806).

Display Mesh...
opens the *Mesh Display Dialog Box* (p. 1635).

Periodic Conditions...
opens the *Periodic Conditions Dialog Box* (p. 1874).

Highlight Zone
when enabled highlights the boundary zone (selected in the task page) in the graphics window.

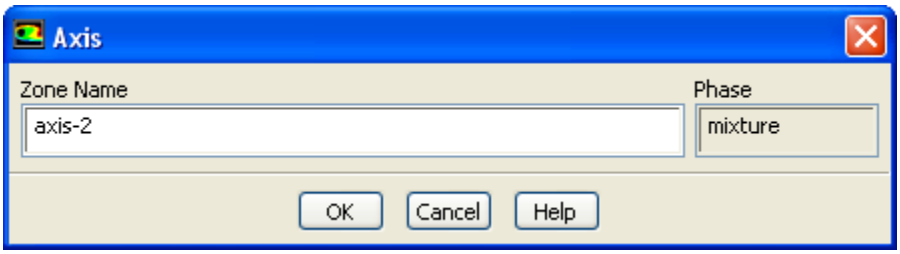
For additional information, please see the following sections:

35.7.1. Axis Dialog Box
35.7.2. Exhaust Fan Dialog Box
35.7.3. Fan Dialog Box
35.7.4. Inlet Vent Dialog Box
35.7.5. Intake Fan Dialog Box
35.7.6. Interface Dialog Box
35.7.7. Interior Dialog Box
35.7.8. Mass-Flow Inlet Dialog Box
35.7.9. Outflow Dialog Box
35.7.10. Outlet Vent Dialog Box
35.7.11. Periodic Dialog Box
35.7.12. Porous Jump Dialog Box
35.7.13. Pressure Far-Field Dialog Box
35.7.14. Pressure Inlet Dialog Box
35.7.15. Pressure Outlet Dialog Box
35.7.16. Radiator Dialog Box
35.7.17. RANS/LES Interface Dialog Box
35.7.18. Symmetry Dialog Box
35.7.19. Velocity Inlet Dialog Box
35.7.20. Wall Dialog Box
35.7.21. Periodic Conditions Dialog Box

35.7.1. Axis Dialog Box

The **Axis** dialog box can be used to modify the name of an axis zone; there are no conditions to be set. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Axis Boundary Conditions* (p. 323) for information about axis boundaries.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



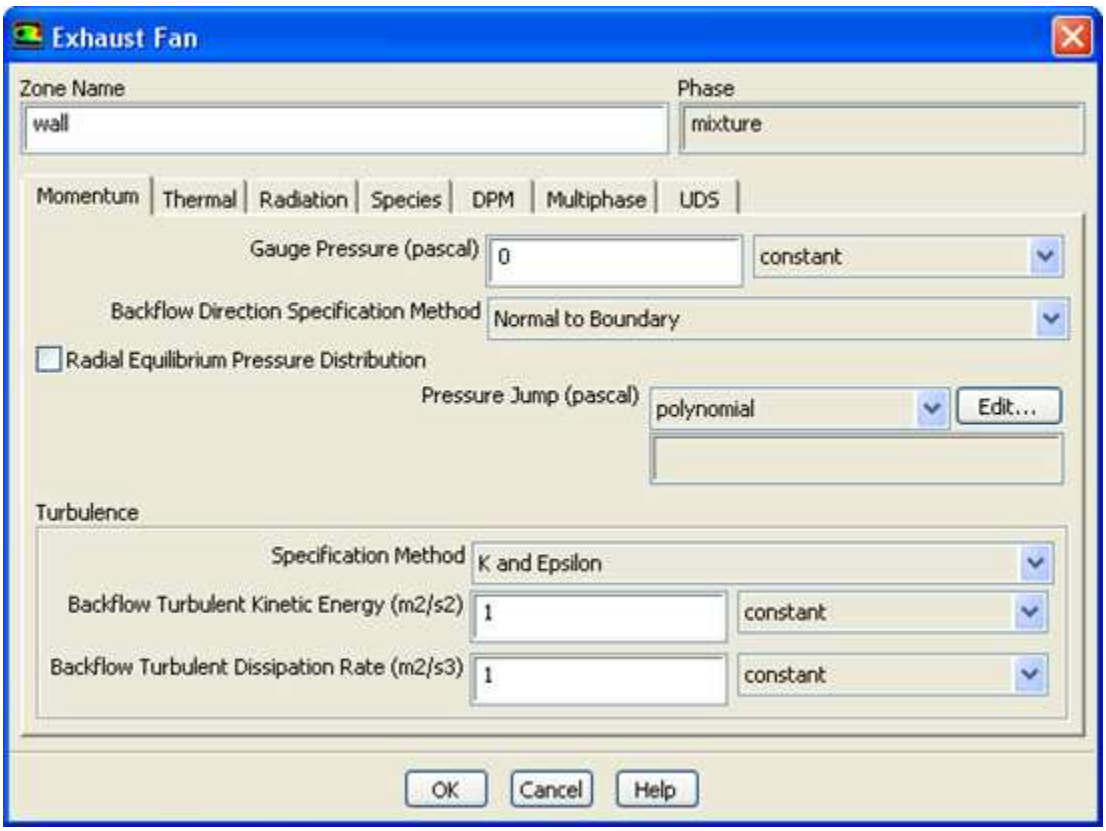
Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

35.7.2. Exhaust Fan Dialog Box

The **Exhaust Fan** dialog box sets the boundary conditions for an exhaust fan zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Exhaust Fan Boundaries* (p. 299) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. It appears only for multiphase flows.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Momentum

contains the momentum parameters.

Gauge Pressure

sets the gauge pressure at the outlet boundary.

Backflow Direction Specification Method

sets the direction of the inflow stream should the flow reverse direction. You can choose **Direction Vector**, **Normal to Boundary**,or **From Neighboring Cell**.

Coordinate System

contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only for **Direction Vector**.

X, Y, Z-Component of Flow Direction

allows you to specify the velocity components in x, y, and z directions respectively. This option is available for cartesian coordinate system.

Radial Equilibrium Pressure Distribution

enables the radial equilibrium pressure distribution. See *Defining Static Pressure* (p. 283) for details.

This item appears only for 3D and axisymmetric swirl solvers.

Pressure Jump

specifies the rise in pressure across the fan. See *Specifying the Pressure Jump* (p. 299) for details.

Target Mass Flow Rate

allows you to set mass flow rate as a boundary condition at the outlet.

Turbulence

displays the turbulence parameters.

Specification Method

specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specification Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Turbulent Viscosity Ratio

sets the value of the backflow turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulence Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters. This parameter is available only when the energy equation is turned on.

Backflow Total Temperature

sets the total temperature of the inflow stream should the flow reverse direction.

Radiation

contains the boundary conditions for the radiation model at the exhaust fan.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing

specifies whether or not the fan participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the fan.

Participates in View Factor Calculation

specifies whether or not the fan participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions

contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters. This tab is available only if you have defined at least one injection.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Backflow Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Backflow Volume Fraction

specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user defines scalars are specified.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

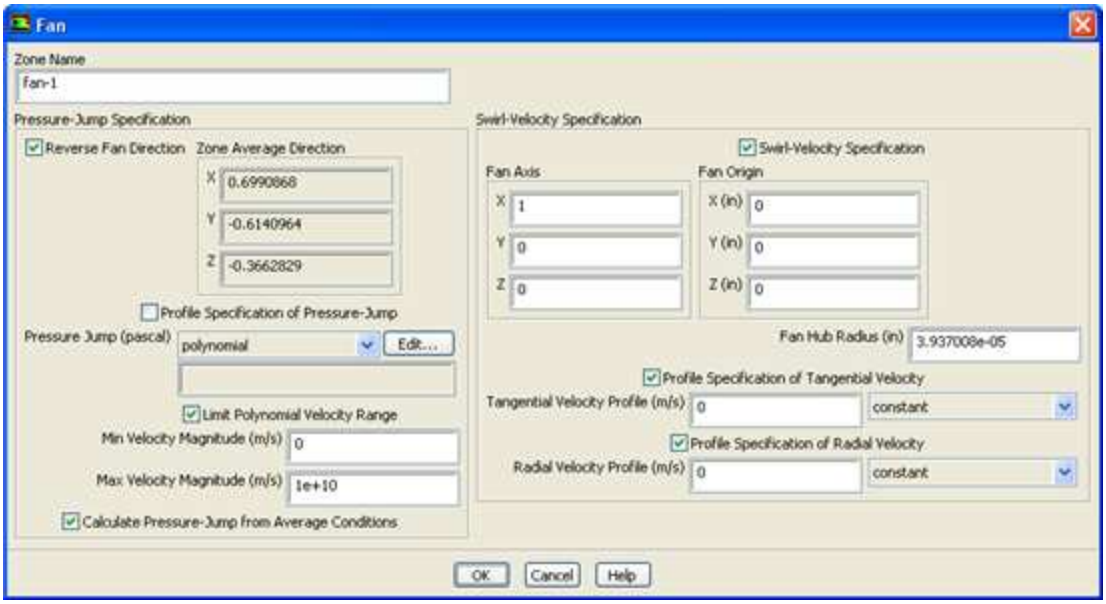
User Scalar-n
specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value
appears only if user defines scalars are specified.

User Scalar-n
specifies the value of the scalar.

35.7.3. Fan Dialog Box

The **Fan** dialog box sets the boundary conditions for a fan zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *User Inputs for Fans* (p. 325) for details about the items below.



Controls

Zone Name
sets the name of the zone.

Pressure-Jump Specification
contains inputs that define the pressure jump across the fan.

Reverse Fan Direction
sets the fan flow direction relative to the zone direction. If **Zone Average Direction** is pointing in the direction you want the fan to blow, do *not* select **Reverse Flow**; if it is pointing in the opposite direction, select **Reverse Flow**.

Zone Average Direction
displays the (face-averaged) direction vector for the zone as an aid in determining whether or not you want to select **Reverse Flow**.

Profile Specification of Pressure-Jump
enables the use of a boundary profile or user-defined function for the pressure jump specification. See *Profiles* (p. 366) or the UDF Manual for details. When this option is enabled, **Pressure Jump Profile** will appear in the dialog box and the next four items below it will not.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure Jump Profile
contains a drop-down list from which you can select a boundary profile or a user-defined function for the pressure jump definition. This item will appear if you enable **Profile Specification of Pressure-Jump**.

Pressure-Jump
specifies the pressure-jump as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See *Defining the Pressure Jump* (p. 326) for details.

Limit Polynomial Velocity Range
limits the minimum and maximum velocity magnitudes used to calculate the pressure jump when it is defined as a function of velocity.

Min Velocity Magnitude, Max Velocity Magnitude
specify the minimum and maximum values to which the velocity magnitude is limited (when the **Limit Polynomial Velocity Range** option is enabled).

Calculate Pressure-Jump from Average Conditions
enables the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior
allows the particles to pass through the boundary.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film
consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Swirl-Velocity Specification
contains inputs for the specification of fan swirl velocity. This section of the dialog box appears only for 3D models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Swirl-Velocity Specification
enables the specification of a swirl velocity for the fan.

Fan Axis
sets the direction vector for the fan's axis of rotation.

Fan Origin
sets the origin in the global coordinate system through which the fan rotation axis passes.

Fan Hub Radius
set the radius of the hub. The default is 1e-6 to avoid division by zero in the polynomial.

Profile Specification of Tangential Velocity
enables the use of a boundary profile or user-defined function for the tangential velocity specification. See *Profiles* (p. 366) or the UDF Manual for details. When this option is enabled, **Tangential Velocity Profile** will appear in the dialog box and **Tangential-Velocity Polynomial Coefficients** will not.

Tangential Velocity Profile
contains a drop-down list from which you can select a boundary profile or a user-defined function for the definition of the tangential velocity. This item will appear if you enable **Profile Specification of Tangential Velocity**.

Tangential-Velocity Polynomial Coefficients
sets the coefficients for the tangential velocity polynomial. Separate the coefficients by spaces.

Profile Specification of Radial Velocity
enables the use of a boundary profile or user-defined function for the radial velocity specification. See *Profiles* (p. 366) or the UDF Manual for details. When this option is enabled, **Radial Velocity Profile** will appear in the dialog box and **Radial-Velocity Polynomial Coefficients** will not.

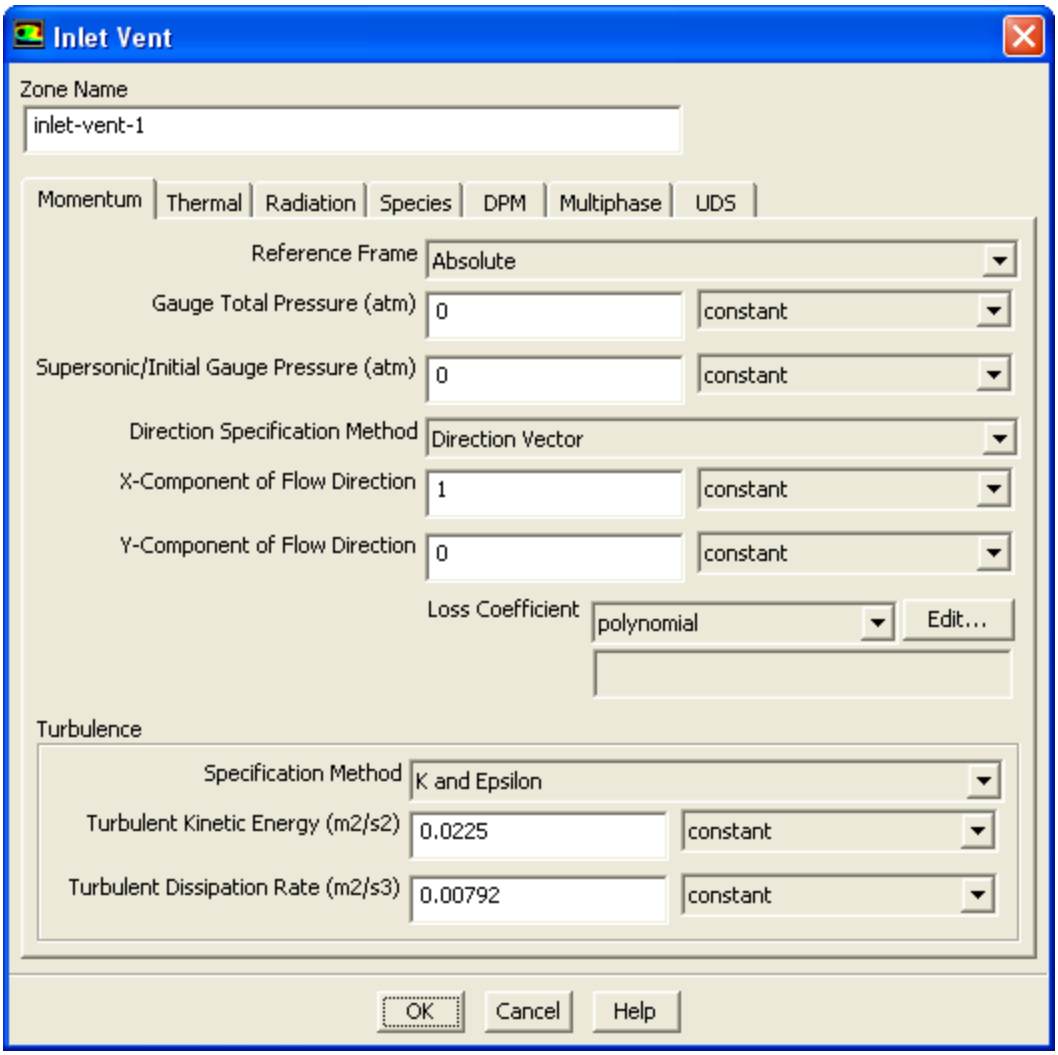
Radial Velocity Profile
contains a drop-down list from which you can select a boundary profile or a user-defined function for the definition of the radial velocity. This item will appear if you enable **Profile Specification of Radial Velocity**.

Radial-Velocity Polynomial Coefficients
sets the coefficients for the radial velocity polynomial. Separate the coefficients by spaces.

35.7.4. Inlet Vent Dialog Box

The **Inlet Vent** dialog box sets the boundary conditions for an inlet vent zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Inlet Vent Boundaries* (p. 278) for details about defining the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Reference Frame

specifies the reference frame for the mass flow. If the cell zone adjacent to the mass-flow inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see *Defining Total Pressure and Temperature* (p. 257) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the inlet vent boundary condition is selected for computing initial values (see *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See *Defining the Flow Direction* (p. 259) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**, **Local Cylindrical Swirl** vector components will be input. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

X,Y,Z-Component of Axis Direction

sets the direction of the axis. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

X,Y,Z-Coordinate of Axis Origin

sets the location of the axis origin. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

Loss-Coefficient

sets the non-dimensional loss coefficient used to compute the pressure drop. See *Specifying the Loss Coefficient* (p. 279)for details.

Turbulence

lists the turbulence parameters.

Specification Method

specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulence Intensity, Turbulence Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulence Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence Intensity, Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Turbulent Viscosity Ratio
sets the value of the turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulence Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Total Temperature
sets the total temperature of the inflow stream. If you are using moving reference frames, see *Defining Total Pressure and Temperature* (p. 257) for information about relative and absolute total temperature.

Radiation
contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing
specifies whether or not the inlet vent participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the inlet vent.

Participates in View Factor Calculation
specifies whether or not the inlet vent participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film
consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction
specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Open Channel
is available when the VOF model with open channel flow is enabled.

Secondary Phase for Inlet
is where the specified parameters are valid only for one secondary phase. In case of a three-phase flow, select the corresponding secondary phase from this list. This appears when **Open Channel** is enabled.

Flow Specification Method
allows you to select the type of flow. You can choose **Free Surface Level and Velocity**, **Total Height and Velocity**, or **Free Surface Level and Total Height**. This appears when **Open Channel** is enabled.

Free Surface Level
can be determined using the absolute value of height from the free surface to the origin in the direction of gravity, or by applying the correct sign based on whether the free surface level is above or below the origin.

Total Height
is used as an option for describing the flow. It is given by *Equation 26–3* (p. 1124).

Bottom Level
is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Velocity Magnitude
sets the magnitude of the velocity vector at the inflow boundary.

Level-Set Function Flux
appears if the **Level Set** option is enabled for the VOF model.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user defines scalars are specified.

User Scalar-n
specifies whether the scalar is a specified flux or a specified value.

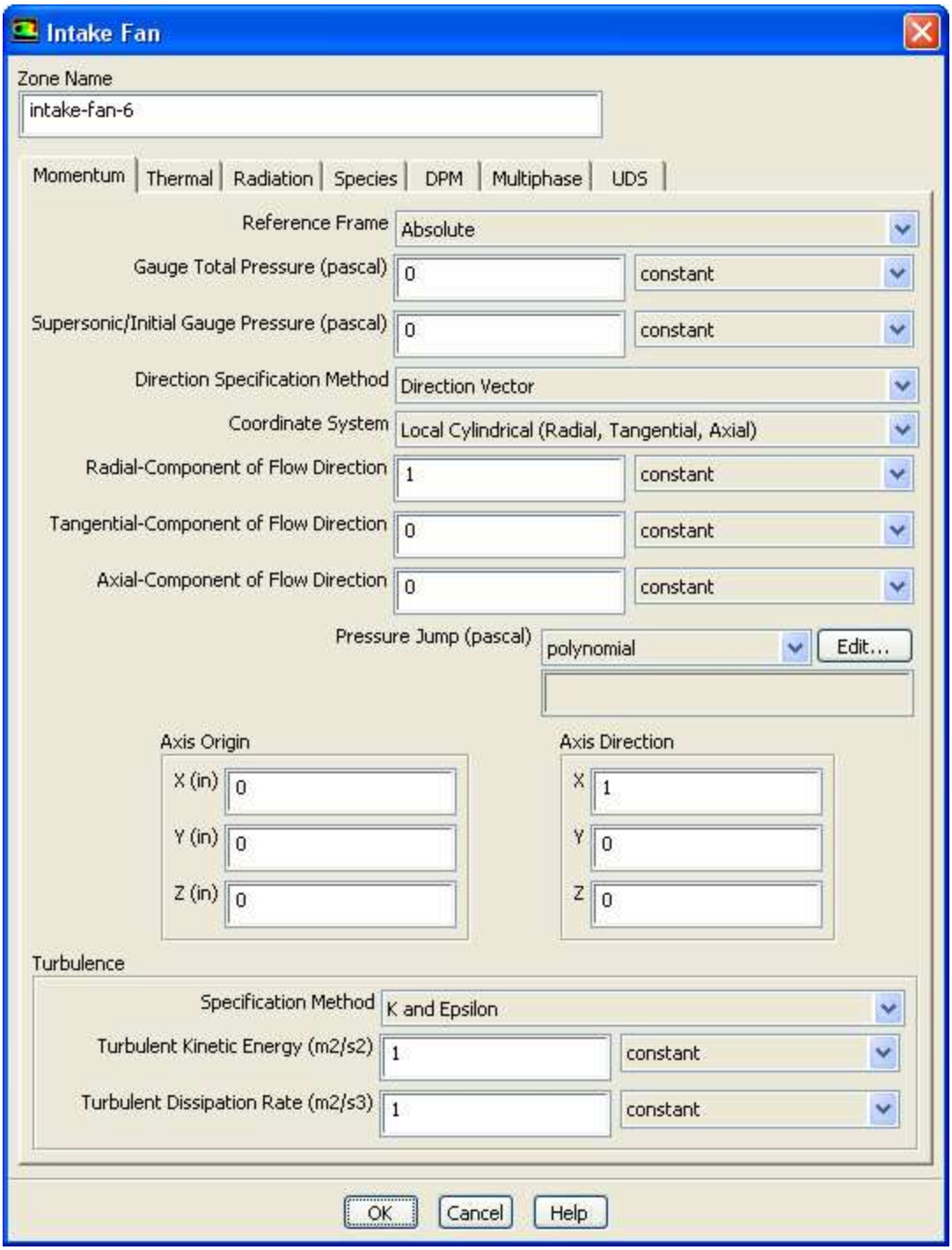
User-Defined Scalar Boundary Value
appears only if user defines scalars are specified.

User Scalar-n
specifies the value of the scalar.

35.7.5. Intake Fan Dialog Box

The **Intake Fan** dialog box sets the boundary conditions for an intake fan zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Intake Fan Boundaries* (p. 280) for details about defining the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Reference Frame

specifies the reference frame for the mass flow. If the cell zone adjacent to the mass-flow inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see *Defining Total Pressure and Temperature* (p. 257) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the intake fan boundary condition is selected for computing initial values (see *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240)).

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See *Defining the Flow Direction* (p. 259) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be input. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction

set the direction of the flow at the inlet boundary. For compressible flow, if the inflow becomes supersonic, the velocity is not reoriented. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction

set the direction of the flow at the inlet boundary. For compressible flow, if the inflow becomes supersonic, the velocity is not reoriented. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Pressure-Jump

specifies the rise in pressure across the fan. See *Specifying the Pressure Jump* (p. 281) for details.

X,Y,Z-Coordinate of Axis Origin

sets the location of the axis origin. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

X,Y,Z-Component of Axis Direction

sets the direction of the axis. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

Turbulence

consists of the turbulence parameters.

Specification Method

specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**.If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components**is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Total Temperature

sets the total temperature of the inflow stream. If you are using moving reference frames, see *Defining Total Pressure and Temperature* (p. 257) for information about relative and absolute total temperature.

Radiation

contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing

specifies whether or not the intake fan participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the intake fan.

Participates in View Factor Calculation

specifies whether or not the intake fan participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as “trapped”. In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having “escaped” when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: "“Wall Jet” Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiphase

contains the multiphase parameters.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction

specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user defines scalars are specified.

User Scalar-n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

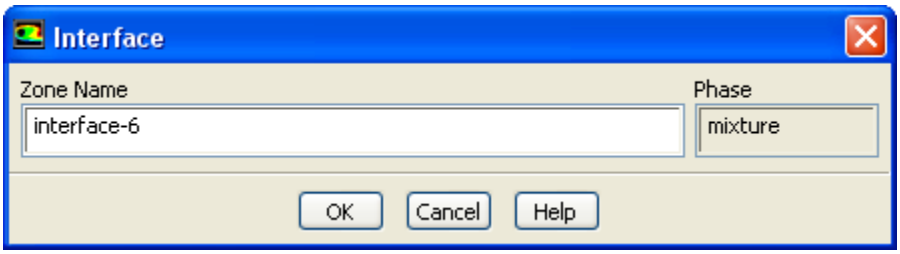
appears only if user defines scalars are specified.

User Scalar-n

specifies the value of the scalar.

35.7.6. Interface Dialog Box

The **Interface** dialog box can be used to modify the name of an interface zone; there are no conditions to be set. It is opened from the *Boundary Conditions Task Page* (p. 1812). Interface zones are used for multiple reference frame and sliding mesh calculations, and for non-conformal meshes. See *The Multiple Reference Frame Model* (p. 522), *Setting Up the Sliding Mesh Problem* (p. 541), and *Non-Conformal Meshes* (p. 161) for details.



Controls

Zone Name

sets the name of the zone.

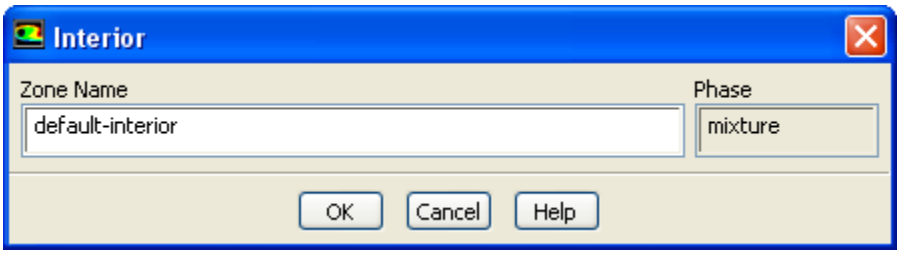
Phase

displays the name of the phase. This item appears only for multiphase flows.

35.7.7. Interior Dialog Box

The **Interior** dialog box can be used to modify the name of an interior zone; there are no conditions to be set. It is opened from the *Boundary Conditions Task Page* (p. 1812).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



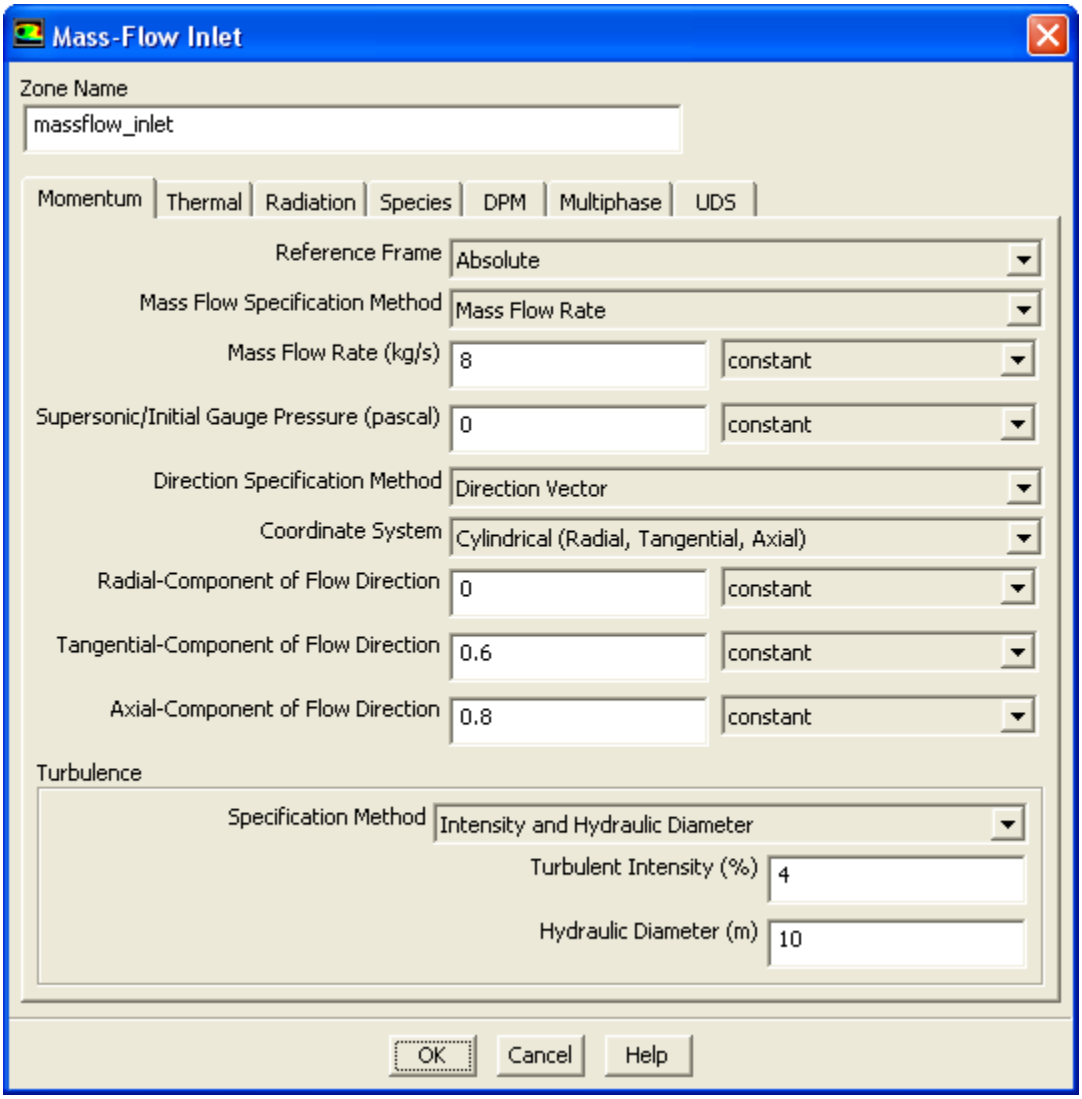
Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears only for multiphase flows.

35.7.8. Mass-Flow Inlet Dialog Box

The **Mass-Flow Inlet** dialog box sets the boundary conditions for a mass-flow inlet zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Mass Flow Inlet Boundaries* (p. 271) for details about defining the items below.



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Zone Name
sets the name of the zone.

Momentum
displays the momentum boundary conditions.

Reference Frame
specifies the reference frame for the mass flow. If the cell zone adjacent to the mass-flow inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Mass Flow Specification Method
specifies whether you are defining **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

Mass Flow Rate
sets the prescribed mass flow rate for the zone. This flow rate is converted internally to a prescribed uniform mass flux over the zone by dividing the flow rate by the flow direction area projection of the zone. This item will appear if you selected **Mass Flow Rate** in the **Mass Flow Specification Method** list.

Important

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

Mass Flux
sets the prescribed mass flux for the zone. This item will appear if you selected **Mass Flux** or **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** list.

Important

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Average Mass Flux
sets the average mass flux through the zone. See *More About Mass Flux and Average Mass Flux* (p. 273) for details. This item will appear if you selected **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** list.

Important

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Supersonic/Initialization Gauge Pressure
sets the static pressure that will be used to initialize the flow field if the mass flow inlet boundary condition is selected for initializing flow properties (see *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See *Defining the Flow Direction* (p. 259) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be input. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow-Direction

set the velocity-direction vector of the inflow stream. This vector does not need to be normalized (e.g., you can specify the vector (1 1 1) rather than (0.577 0.577 0.577)). These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction

set the velocity-direction vector of the inflow stream. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Radial-, Axial-Component of Flow Direction and Tangential-Component of Velocity

appear for a 3D **Local Cylindrical Swirl** coordinate system. Specify the **X**, **Y**, and **Z-Component of Axis Direction** and the **X**, **Y**, and **Z-Coordinate of Axis Origin**.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components**is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Total Temperature
sets the total temperature of the inflow stream.

Radiation
contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing
specifies whether or not the mass-flow inlet participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the mass-flow inlet.

Participates in View Factor Calculation
specifies whether or not the mass-flow inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film
consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user defines scalars are specified.

User Scalar-n
specifies whether the scalar is a specified flux or a specified value.

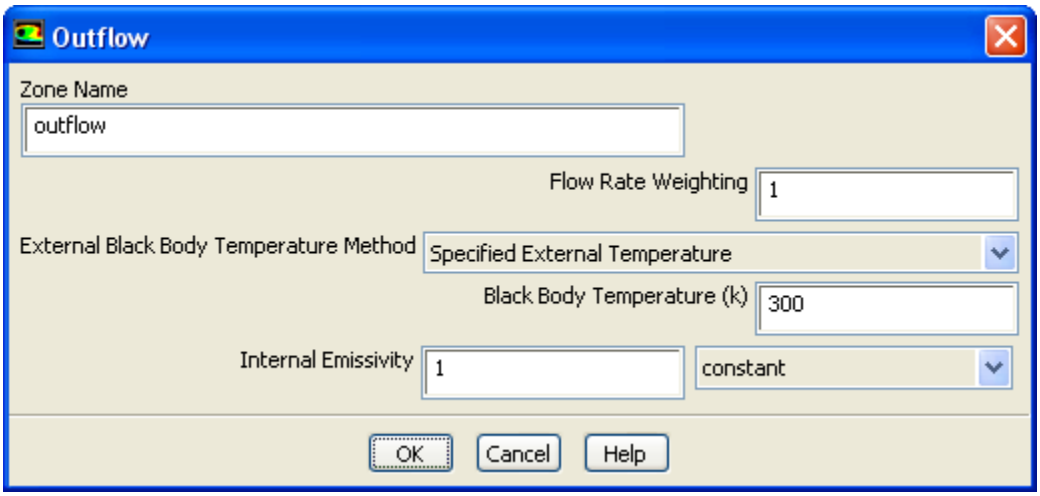
User-Defined Scalar Boundary Value
appears only if user defines scalars are specified.

User Scalar-n
specifies the value of the scalar.

35.7.9. Outflow Dialog Box

The **Outflow** dialog box sets the boundary conditions for an outflow zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Using Outflow Boundaries* (p. 294) for details about using outflow boundaries.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Flow Rate Weighting

specifies the portion of the outflow that is going through the boundary. See *Mass Flow Split Boundary Conditions* (p. 295) for details.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates in Solar Ray Tracing
specifies whether or not outflow participate in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the outflow.

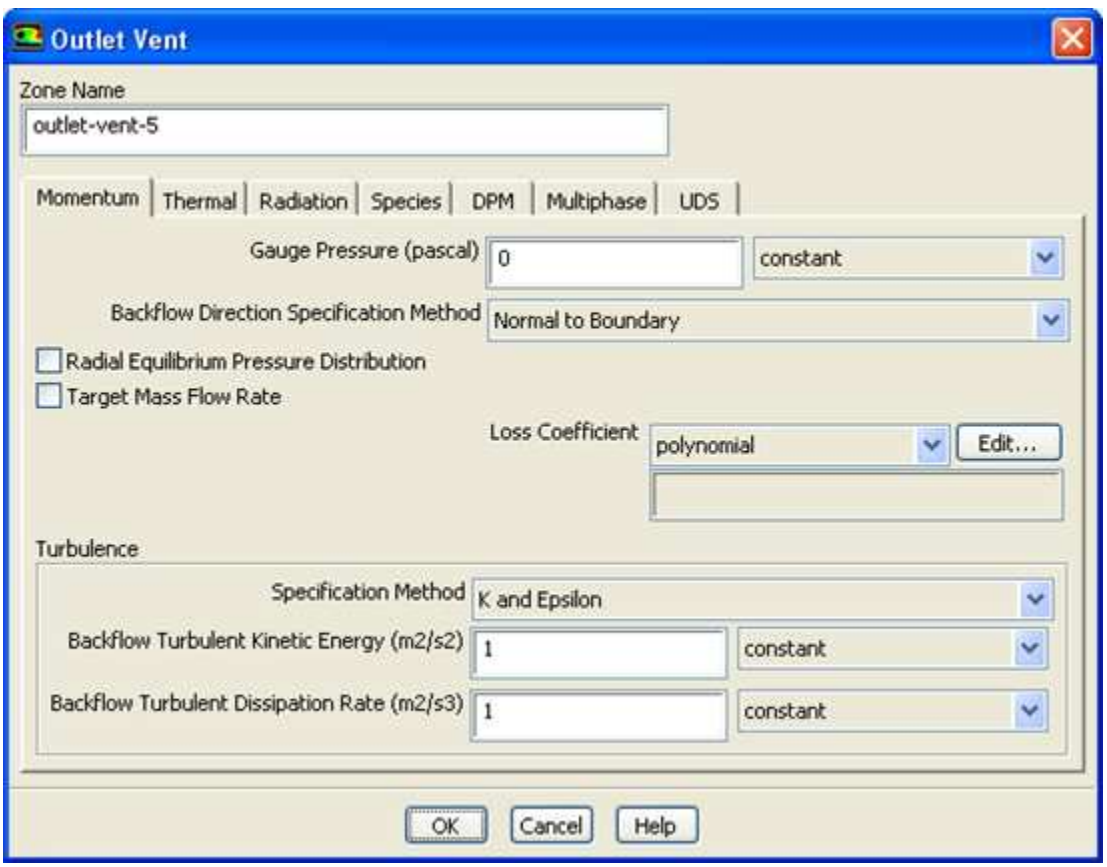
Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Participates in View Factor Calculation

specifies whether or not the outflow participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

35.7.10. Outlet Vent Dialog Box

The **Outlet Vent** dialog box sets the boundary conditions for an outlet vent zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Outlet Vent Boundaries* (p. 297) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Momentum
contains the momentum parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gauge Pressure
sets the gauge pressure at the outlet boundary.

Radial Equilibrium Pressure Distribution
enables the radial equilibrium pressure distribution. See *Defining Static Pressure* (p. 283) for details.

This item appears only for 3D and axisymmetric swirl solvers.

Backflow Direction Specification Method
specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** or **From Neighboring Cell** no inputs are required. See *Defining the Flow Direction* (p. 259) for information on specifying flow direction.

Target Mass Flow Rate
allows you to set mass flow rate as a boundary condition at the outlet.

Loss-Coefficient
sets the non-dimensional loss coefficient used to compute the pressure drop. See *Specifying the Loss Coefficient* (p. 298) for details.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio
sets the value of the backflow turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Intensity or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Backflow Total Temperature
sets the total temperature of the inflow stream should the flow reverse direction

Radiation
contains the boundary conditions for the radiation model at the outlet vent.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing
specifies whether or not the outlet vent participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the outlet vent.

Participates in View Factor Calculation
specifies whether or not the outlet vent participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions
contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Backflow Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Backflow Volume Fraction

specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user defines scalars are specified.

User Scalar-n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user defines scalars are specified.

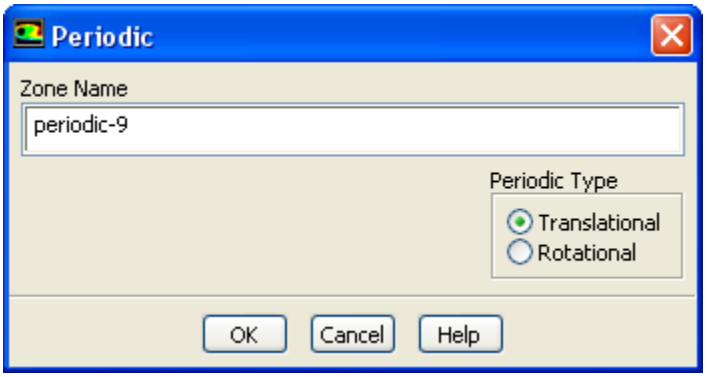
User Scalar-n

specifies the value of the scalar.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.7.11. Periodic Dialog Box

The **Periodic** dialog box sets the boundary conditions for a periodic zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs for Periodic Boundaries* (p. 321) for details about the items below. See *Periodic Flows* (p. 493) for information about fully-developed periodic flow.



Controls

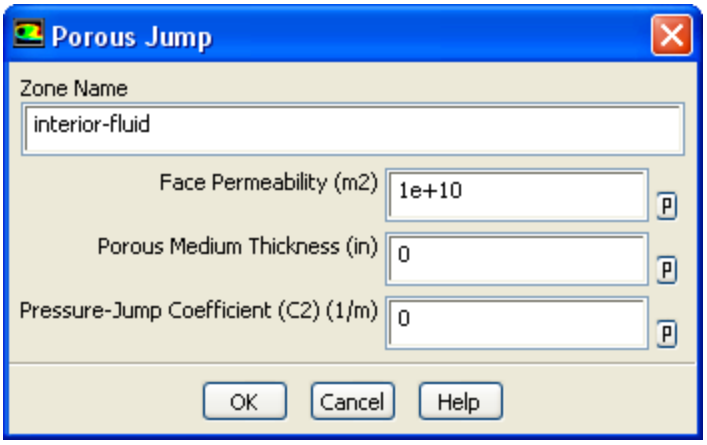
Zone Name
sets the name of the zone.

Periodic Type
indicates whether the periodicity of the domainis **Translational** or **Rotational**.

Periodic Pressure Jump
sets the pressure increase/decrease across the periodic boundary. (This item will not appear if the pressure-based (default) solver is used; it is relevant only for the density-based solvers.)

35.7.12. Porous Jump Dialog Box

The **Porous Jump** dialog box sets the boundary conditions for a porous-jump zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Porous Jump Boundary Conditions* (p. 337) for details about the items below.



Controls

Zone Name
sets the name of the zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Face Permeability

sets the face permeability coefficient (α in *Equation 7–112* (p. 337)).

This quantity can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Porous Medium Thickness

sets the thickness of the porous medium (Δm).

This quantity can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Pressure-Jump Coefficient

sets the pressure-jump coefficient (C_2).

This quantity can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Jump Adhesion

sets the adhesion method and contact angle.

Constrained-Two-Sided Adhesion

constrains the contact angle at the porous jump, When this option is disabled then the forced two-sided adhesion treatment is in effect. See Jump Adhesion for more information.

Contact Angle

is the contact angle at the porous jump.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior

allows the particles to pass through the boundary.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Solar Boundary Conditions

contains the settings for solar ray tracing. This group box is available only if you select **Solar Ray Tracing** from the **Model** list in the **Solar Load** group box of the **Radiation Model** dialog box. See *Solar Ray Tracing* (p. 754) for details.

Participates in Solar Ray Tracing
specifies whether or not the porous jump participates in solar ray tracing.

Absorptivity
contains the settings that define the absorptivity of the porous jump.

Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the absorption of the porous jump.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the absorption of the porous jump.

Transmissivity
contains the settings that define the transmissivity of the porous jump.

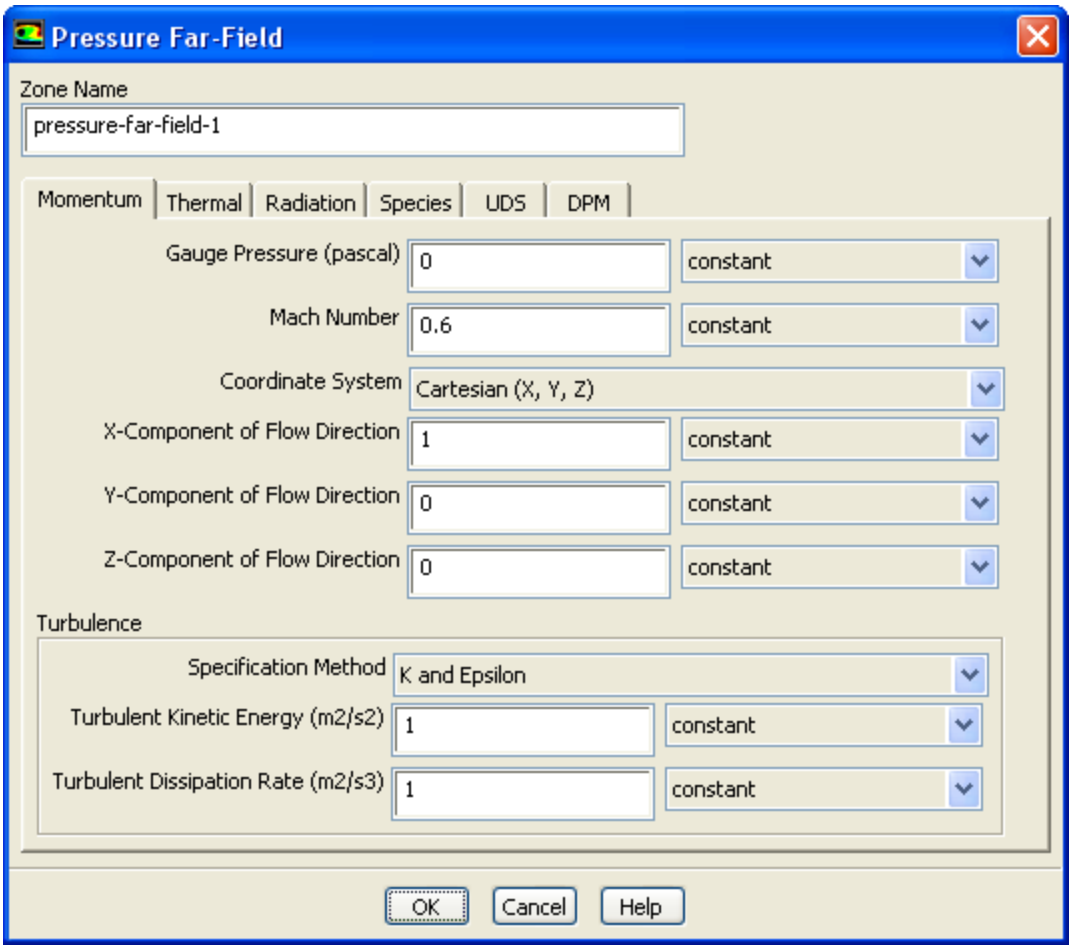
Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the transmissivity of the porous jump.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the transmissivity of the porous jump.

35.7.13. Pressure Far-Field Dialog Box

The **Pressure Far-Field** dialog box sets the boundary conditions for a pressure far-field zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Pressure Far-Field Boundaries* (p. 291) for details about defining the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Gauge Pressure

sets the far-field gauge static pressure.

Mach Number

sets the far-field Mach number. The Mach number can be subsonic, sonic, or supersonic.

Coordinate System

allows you to select a **Cartesian**, **Cylindrical**, or **Local Cylindrical** coordinate system. This option is available only for 3D geometry.

X,Y,Z-Component of Flow-Direction

set the far-field flow direction. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction

set the far-field flow direction. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**. Specify the **X**, **Y**, and **Z-Component of Axis Direction** and the **X**, **Y**, and **Z-Coordinate of Axis Origin** for the **Local Cylindrical** coordinate system.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Turbulence Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components**is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Temperature

sets the far-field static temperature.

Radiation

contains the boundary conditions for the radiation model at the pressure far-field zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing
specifies whether or not the pressure far-field zone participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure far-field zone.

Participates in View Factor Calculation
specifies whether or not the pressure far-field zone participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user defines scalars are specified.

User Scalar-n
specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value
appears only if user defines scalars are specified.

User Scalar-n
specifies the value of the scalar.

DPM
contains the discrete phase parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

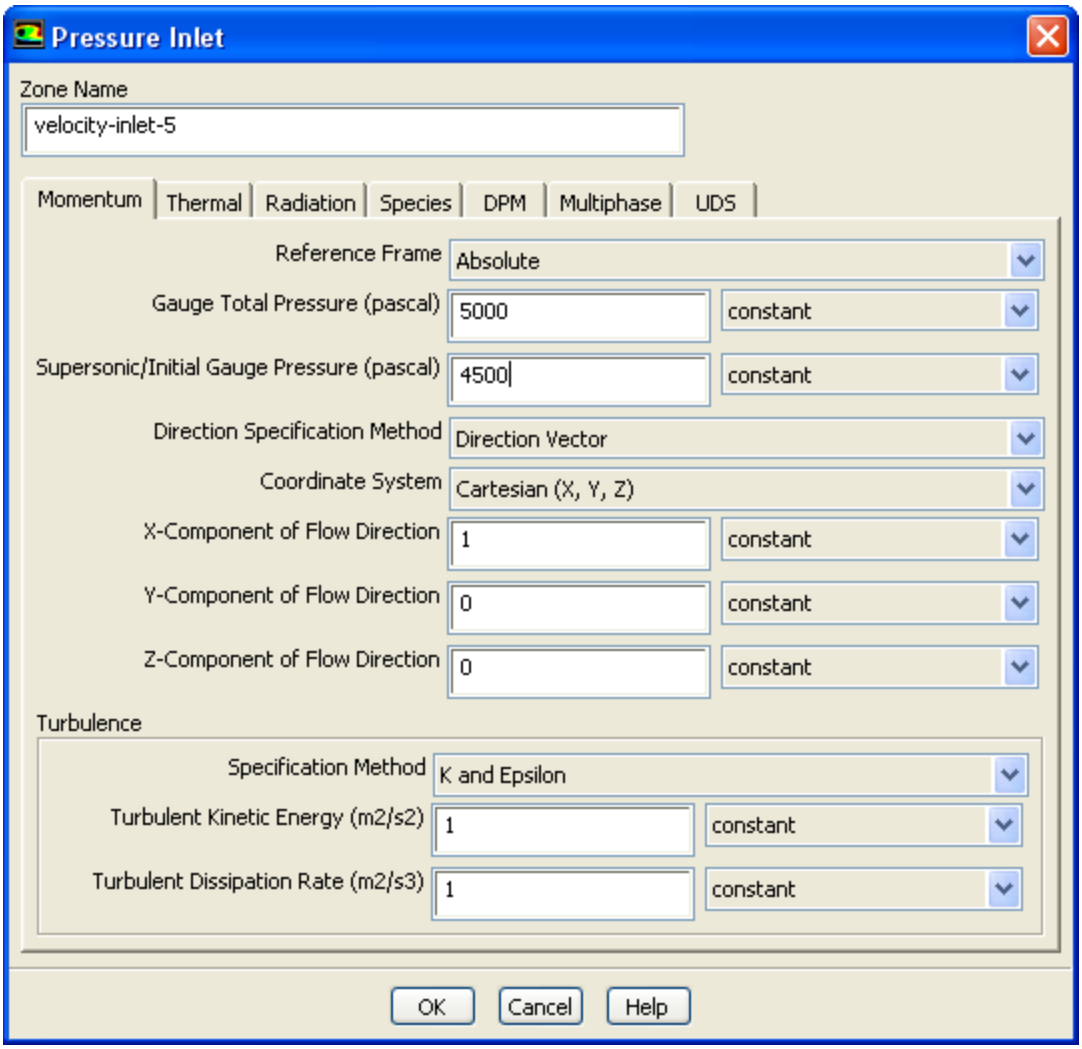
Discrete Phase BC Function

sets the user-defined function from the drop-down list.

35.7.14. Pressure Inlet Dialog Box

The **Pressure Inlet** dialog box sets the boundary conditions for a pressure inlet zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Pressure Inlet Boundaries* (p. 255) for details about defining the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Reference Frame

specifies the reference frame for the pressure inlet. If the cell zone adjacent to the pressure inlet is moving, you can choose to specify the total temperature, total pressure, and velocity components as **Relative to Adjacent CellZone**or **Absolute**in the **Reference Frame** drop-down list.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see *Defining Total Pressure and Temperature* (p. 257) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the pressure inlet boundary condition is selected for computing initial values (see *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240)).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See *Defining the Flow Direction* (p. 259) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be input. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

X,Y,Z-Component of Axis Direction

sets the direction of the axis. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

X,Y,Z-Coordinate of Axis Origin

sets the location of the axis origin. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

Radial-, Axial-Component of Flow Direction and Tangential-Component of Velocity

appear for a 3D **Local Cylindrical Swirl** coordinate system. Specify the **X**, **Y**, and **Z-Component of Axis Direction** and the **X**, **Y**, and **Z-Coordinate of Axis Origin**.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components**is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Total Temperature

sets the total temperature of the inflow stream. If you are using moving reference frames, see *Defining Total Pressure and Temperature* (p. 257) for information about relative and absolute total temperature.

Radiation

contains the boundary conditions for the radiation model at the pressure inlet.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing

specifies whether or not the pressure inlet participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure inlet.

Participates in View Factor Calculation

specifies whether or not the pressure inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film
consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction
specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user defines scalars are specified.

User Scalar-n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

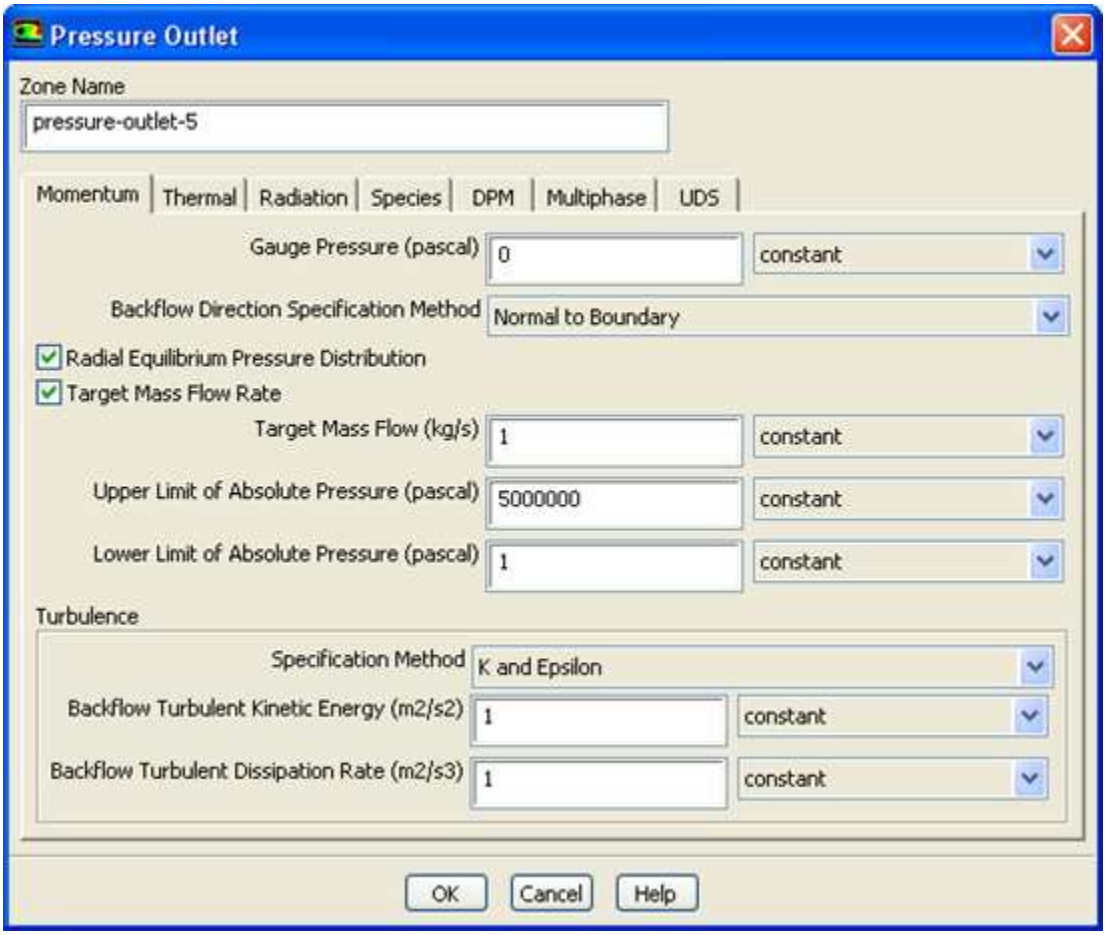
appears only if user defines scalars are specified.

User Scalar-n

specifies the value of the scalar.

35.7.15. Pressure Outlet Dialog Box

The **Pressure Outlet** dialog box sets the boundary conditions for a pressure outlet zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Pressure Outlet Boundaries* (p. 282) for details about defining the items below.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gauge Pressure
sets the gauge pressure at the outflow boundary.

Backflow Direction Specification Method
sets the direction of the inflow stream should the flow reverse direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** or **From Neighboring Cell**, no inputs are required. See *Inputs at Pressure Outlet Boundaries* (p. 282) for information on specifying flow direction.

Radial Equilibrium Pressure Distribution
enables the radial equilibrium pressure distribution. See *Defining Static Pressure* (p. 283)for details.

This item appears only for 3D and axisymmetric swirl solvers.

Average Pressure Specification
allows the pressure along the outlet boundary to vary, but maintain an average equivalent to the specified value in the **Gauge Pressure** input field. In this boundary implementation, the pressure variation provides a low level of non-reflectivity. For more details, see*Calculation Procedure at Pressure Outlet Boundaries* (p. 285).

Note

The **Average Pressure Specification** option is not available if the **Radial Equilibrium Pressure Distribution** option is enabled.

Target Mass Flow Rate
allows you to set mass flow rate as a boundary condition at the outlet.

Target Mass Flow
allows you to specify the flow as either a constant value or a UDF.

Upper Limit of Absolute Pressure, Lower Limit of Absolute Pressure
specifies the range of the pressure limits, which have different pressure variations on different boundaries. The upper and lower pressure limits can be specified as a constant or a profile.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio
sets the value of the backflow turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Backflow Total Temperature
sets the total temperature of the inflow stream should the flow reverse direction

Radiation
contains the boundary conditions for the radiation model at the pressure outlet.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing
specifies whether or not the pressure outlet participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure outlet.

Participates in View Factor Calculation
specifies whether or not the pressure outlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions
contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film
consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Backflow Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Volume Fraction
specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user defines scalars are specified.

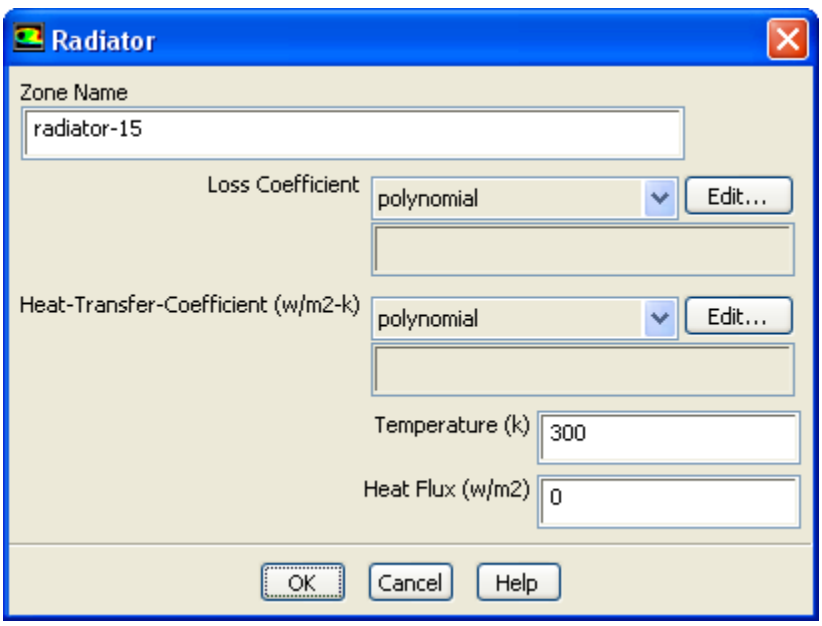
User Scalar-n
specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value
appears only if user defines scalars are specified.

User Scalar-n
specifies the value of the scalar.

35.7.16. Radiator Dialog Box

The **Radiator** dialog box sets the boundary conditions for a radiator model zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *User Inputs for Radiators* (p. 332) for details about the items below.



Controls

Zone Name
sets the name of the zone.

Loss Coefficient
specifies the loss coefficient as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See *Defining the Pressure Loss Coefficient Function* (p. 333) for details.

Heat-Transfer-Coefficient
specifies the heat-transfer coefficient as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See *Defining the Heat Flux Parameters* (p. 335) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature
sets the temperature used to compute heat flux from the radiator using the **Heat-Transfer-Coefficient**. If **Temperature**is zero, the **Heat Flux** condition is used instead.

Heat Flux
sets the heat flux at the radiator surface (used only when **Temperature** is zero).

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior
allows the particles to pass through the boundary.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film
consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

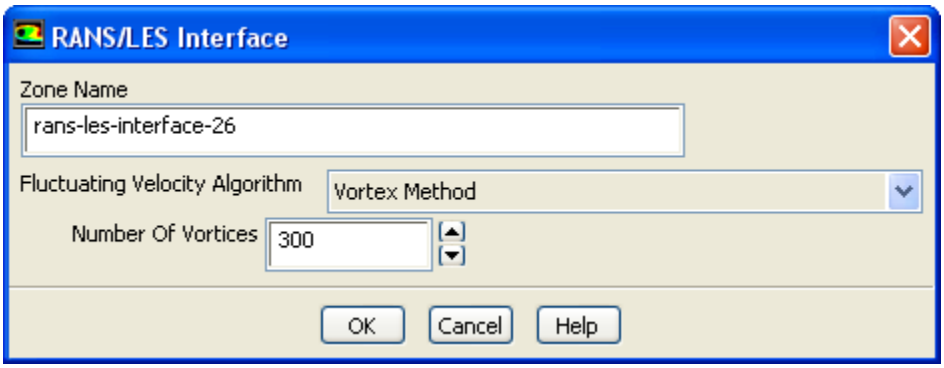
user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

35.7.17. RANS/LES Interface Dialog Box

The **RANS/LES Interface** dialog box can be used to create artificial resolved turbulence (fluctuation/perturbations) at the interface where the flow proceeds from the RANS zone into the LES zone of the computational domain for Embedded LES turbulent flows. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Setting Up the Embedded Large Eddy Simulation (E-LES) Model* (p. 670) for more information about RANS/LES interfaces.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name
sets the name of the RANS/LES interface.

Fluctuating Velocity Algorithm
displays the methods for generating fluctuating velocity components at the RANS/LES interface. Available options include:

- No Perturbations
- Spectral Synthesizer
- Vortex Method

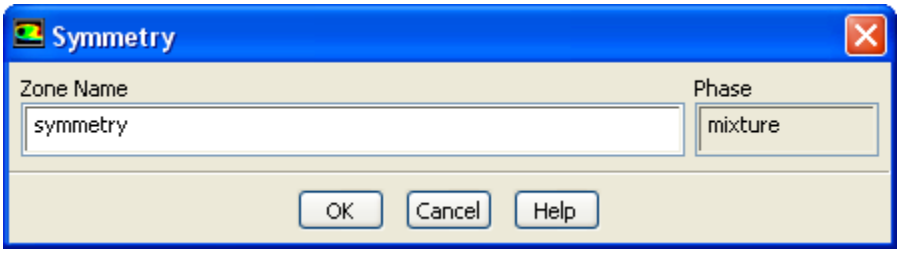
Number of Vortices
displays the amount of vortices that the selected fluctuating velocity method distributes randomly over the face zone and uses to generate turbulent fluctuations (available for the Vortex Method only).

Important

The RANS/LES 'interface' can either be an interior zone (that has been assigned to be a **rans-les-interface** zone), or it can be a non-conformal interface. If it is a non-conformal interface, the you need to identify the name of the non-conformal interface's "interior" zone (e.g., using the **Mesh Interfaces** task page), and then go to that zone in the **Boundary Conditions** task page.

35.7.18. Symmetry Dialog Box

The **Symmetry** dialog box can be used to modify the name of a symmetry zone; there are no conditions to be set. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Symmetry Boundary Conditions* (p. 319) for information about symmetry boundaries.



Controls

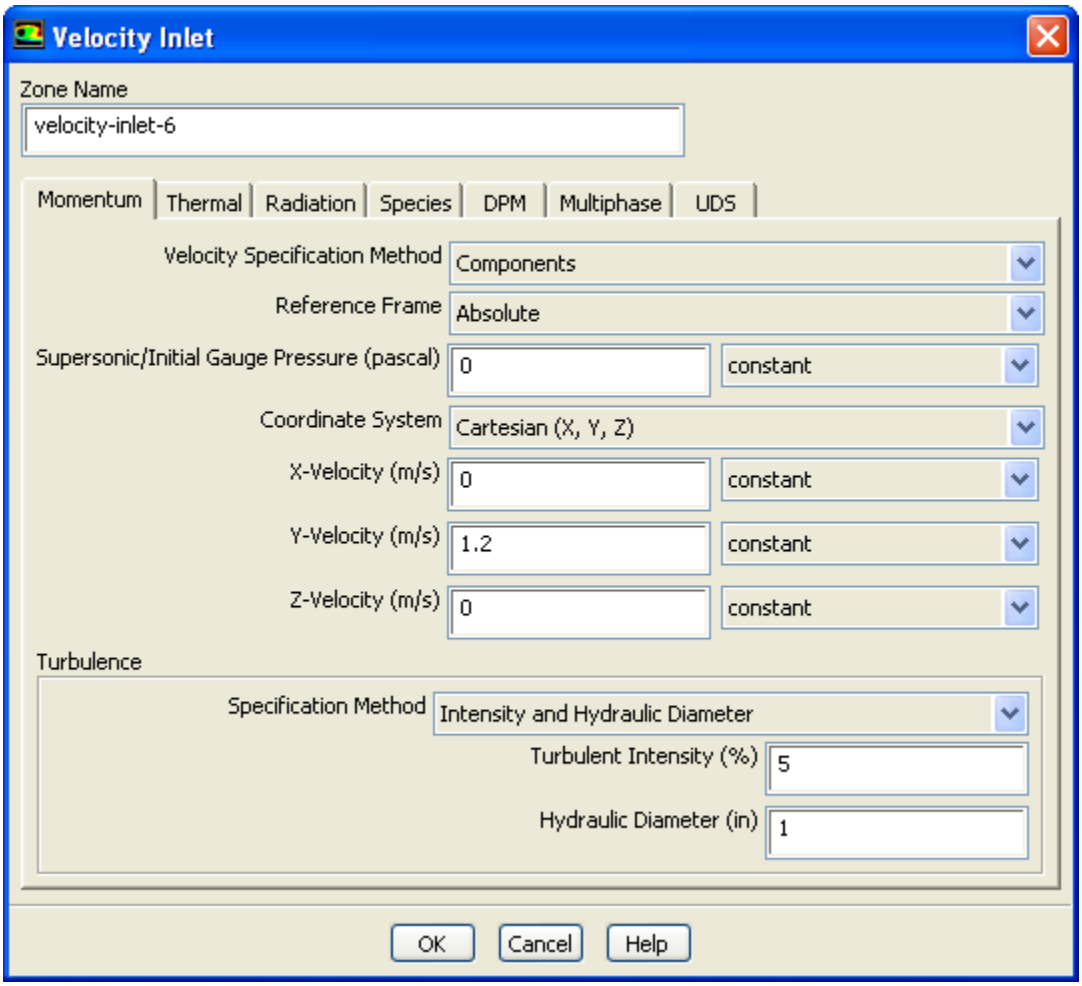
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item is available only for multiphase flows.

35.7.19. Velocity Inlet Dialog Box

The **Velocity Inlet** dialog box sets the boundary conditions for a velocity inlet zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Velocity Inlet Boundaries* (p. 265) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

Open Channel Wave BC
allows you to set specific parameters for a particular boundary for open channel wave boundaries. This is available when the volume of fluid multiphase model is selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Momentum

contains the momentum parameters.

Velocity Specification Method

sets the method used to define the inflow velocity.

Flow Direction Specification Method

sets the method used to define the direction of flow of the wave. This is available when you enable the **Open Channel Wave BC** option.

Magnitude and Direction

allows specification in terms of a **Velocity Magnitude** and **Flow-Direction**.

Components

allows specification in terms of the Cartesian, cylindrical, or local cylindrical velocity components.

Magnitude, Normal to Boundary

allows specification of a **Velocity Magnitude** normal to the boundary.

Reference Frame

specifies relative or absolute velocity inputs. You can choose to enter **Absolute** velocities or velocities **Relative to Adjacent Cell Zone**. If you are not using moving reference frames, both options are equivalent, so you need not choose.

Uniform Flow Velocity Magnitude

is the flow velocity, specified as a constant or a parameter.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, or **Local Cylindrical** velocities will be input. This item will appear only for 3D cases in which you have selected **Magnitude and Direction** or **Components** as the **Velocity Specification Method**.

X,Y,Z-Velocity

set the components of the velocity vector at the inflow boundary. These items will appear for 2D non-axisymmetric models, or for 3D models if you select the **Components** option as the **Velocity Specification Method** and **Cartesian** as the **Coordinate System**.

Radial, Tangential, Axial-Velocity

set the components of the velocity vector at the inflow boundary. These items will appear for 3D models if you select the **Components** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**.

Axial, Radial, Swirl-Velocity

set the components of the velocity vector at the inflow boundary. These items will appear for 2D axisymmetric models.

Important

Swirl-Velocity will appear only for 2D axisymmetric swirl models.

Angular Velocity

specifies the angular velocity Ω for a 3D flow. This item will appear for a 3D model if you select the **Components** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**.

Swirl Angular Velocity

specifies the swirl angular velocity Ω for an axisymmetric swirling flow. This item will appear for an axisymmetric swirl model if you choose **Components** as the **Velocity Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocity Magnitude
sets the magnitude of the velocity vector at the inflow boundary. This item will appear if you select the **Magnitude and Direction** or **Magnitude, Normal to Boundary** option as the **Velocity Specification Method**.

X,Y,Z-Component of Flow-Direction
set the direction of the velocity vector at the inflow boundary. These items will appear for 2D non-axisymmetric models if you select the **Magnitude and Direction** option as the **Velocity Specification Method**, or for 3D models if you select the **Magnitude and Direction** option as the **Velocity Specification Method** and **Cartesian** as the **Coordinate System**.

Radial, Tangential, Axial-Component of Flow Direction
set the direction of the velocity vector at the inlet boundary. These items will appear for 3D models if you select the **Magnitude and Direction** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**, or for 2D axisymmetric models.

Important

Tangential-Velocity will appear only for 2D axisymmetric swirl models.

X,Y,Z-Component of Axis Direction
sets the direction of the axis. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

X,Y,Z-Coordinate of Axis Origin
sets the location of the axis origin. These items will appear if the selected **Coordinate System** is **Local Cylindrical**.

Outflow Gauge Pressure
specifies the pressure to be used as the pressure outlet condition if flow exits the domain at any face on the velocity inlet boundary. (Note that this effect is similar to that of the "velocity far-field" boundary that was available in RAMPANT 3.)

This item appears only for the density-based solvers.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to input the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See *Determining Turbulence Parameters* (p. 250) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t/μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t/μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity

sets the value of the turbulence intensity I for the LES model.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS FLUENT will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See *Reynolds Stress Model* (p. 679) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components**is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Temperature

specifies the static temperature of the flow.

Radiation

contains the boundary conditions for the radiation model at the velocity inlet.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1 model, the DTRM, the discrete ordinates model, or the S2S model for radiation heat transfer. See *Defining Boundary Conditions for Radiation* (p. 724) for details.

Participates in Solar Ray Tracing

specifies whether or not the velocity inlet participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the velocity inlet.

Participates in View Factor Calculation

specifies whether or not the velocity inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See *Defining Cell Zone and Boundary Conditions for Species* (p. 822) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. These items will appear only if you are using the non-premixed or partially premixed combustion model.

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See *Setting Boundary Conditions for the Progress Variable* (p. 899) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See *Figure 25.19* (p. 1051))

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.2: ""Wall Jet" Boundary Condition for the Discrete Phase" in the Theory Guide.

wall-film

consists of four regimes: stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. Detailed information on the wall-film model can be found in Wall-Film Model Theory in the Theory Guide. The **Number Of Splashed Drops** must be specified.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume Fraction
specifies the volume fraction of the secondary phase selected in the *Boundary Conditions Task Page* (p. 1812). This section of the dialog box will appear when one of the multiphase models is being used. See *Defining Multiphase Cell Zone and Boundary Conditions* (p. 1107) for details.

Wave Theory
allows you to choose from **First Order Airy** (the default), **Second Order Stokes**, **Third Order Stokes**,**Fourth Order Stokes**, and **Fifth Order Stokes**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave BC Options
allows you to choose between **Shallow/Intermediate Waves** or **Short Gravity Waves**. Information about the two types of waves is available in Backflow Volume Fraction Specification in the Theory Guide.

Secondary Phase for Inlet
is where the specified parameters are valid only for one secondary phase. In case of a three-phase flow, select the corresponding secondary phase from this list.

Wave Amplitude
is the amplitude of the shallow wave or short gravity wave.

Wave Length
is the wave length of the shallow wave or short gravity wave.

Free Surface Level
can be determined using the absolute value of height from the free surface to the origin in the direction of gravity, or by applying the correct sign based on whether the free surface level is above or below the origin.

Bottom Level
is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Phase Difference
is the phase difference between one wave and another.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user defines scalars are specified.

User Scalar-n
specifies whether the scalar is a specified flux or a specified value.

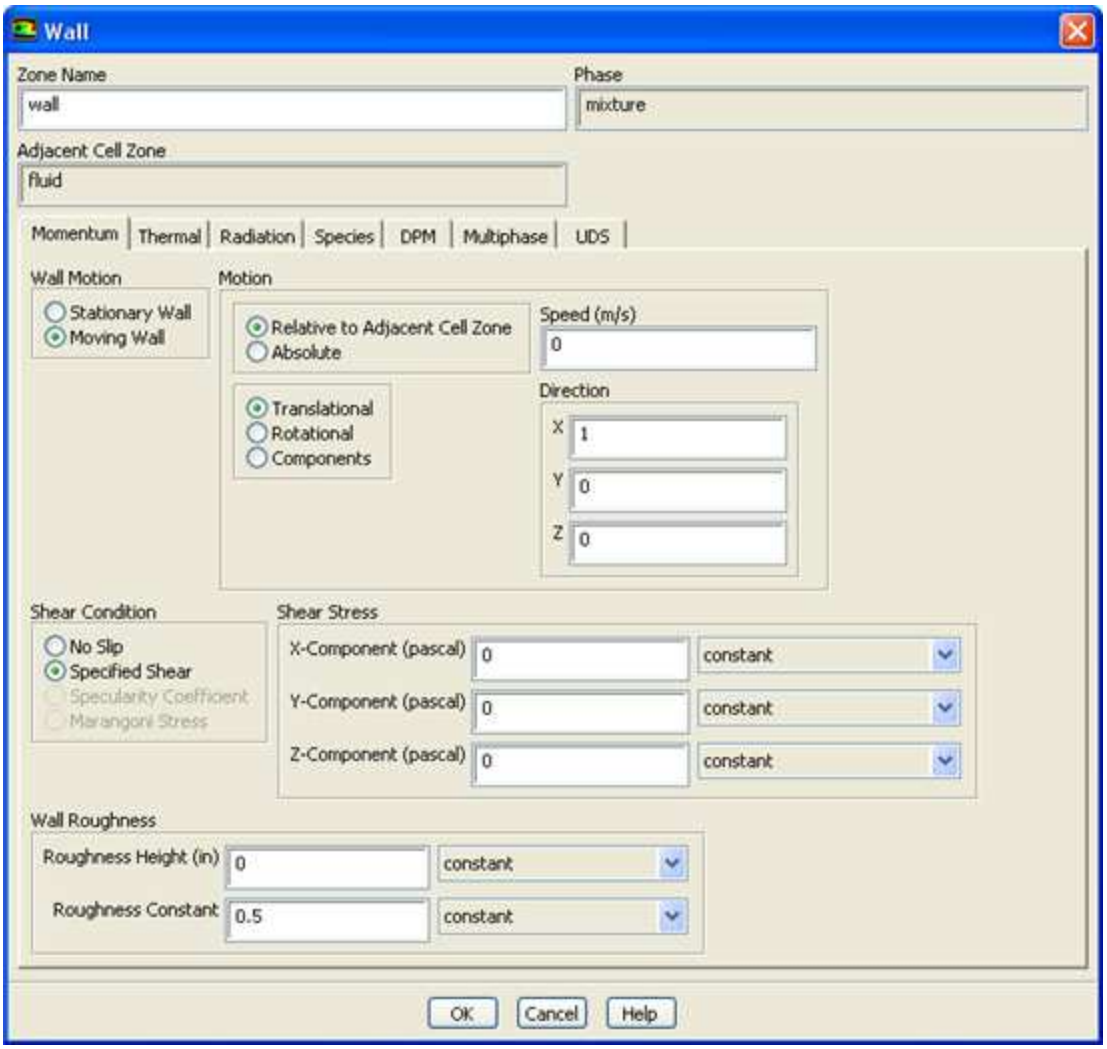
User-Defined Scalar Boundary Value
appears only if user defines scalars are specified.

User Scalar-n
specifies the value of the scalar.

35.7.20. Wall Dialog Box

The **Wall** dialog box sets the boundary conditions for a wall zone. It is opened from the *Boundary Conditions Task Page* (p. 1812). See *Inputs at Wall Boundaries* (p. 300) for details about defining the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

Adjacent Cell Zone

shows the name of the cell zone adjacent to the wall. (This is for informational use only; you cannot edit this field.)

Momentum

displays the momentum boundary conditions.

Wall Motion

contains options for specifying whether or not the wall is moving.

Stationary Wall

specifies that the wall is not moving relative to the adjacent cell zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving Wall

enables specification of the tangential wall motion. Tangential wall motion is applicable only to viscous flows. Since the inviscid slip condition decouples the tangential wall velocity from the governing equations, tangential wall motion has no effect on inviscid flow.

Motion

contains inputs related to wall motion. See *Velocity Conditions for Moving Walls* (p. 302) for details.

Relative to Adjacent Cell Zone

enables the specification of a wall velocity relative to the velocity of the adjacent cell zone.(If the adjacent cell zone is not moving, this is equivalent to **Absolute**.)

Absolute

enables the specification of an absolute wall velocity,

Translational

enables the specification of a translational wall velocity.

Rotational

enables the specification of a rotational wall velocity.

Components

enables the specification of wall velocity components.

Speed

sets the translational or rotational speed of the wall (depending on whether you selected **Translational** or **Rotational**).

The rotational speed can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Direction

sets the direction vector of the translational velocity. (This item will appear if you have chosen the **Translational** option.)

Rotation-Axis Origin

sets the coordinates of the origin of the axis of rotation, thereby determining the location of the axis. (This item will appear if you have chosen the **Rotational** option for a non-axisymmetric case.)

Rotation-Axis Direction

sets the direction vector for the axis of rotation. (This item will appear if you have chosen the **Rotational** option for a non-axisymmetric case.)

Velocity Components

sets the **X**, **Y**, and **Z-Velocity**components of the wall motion. (This item will appear if you have chosen the **Components** option.)

Shear Condition

contains options for specifying the shear conditions at the wall.

No Slip

specifies a no-slip condition at the wall. No further inputs are required.

Specified Shear

enables specification of zero or non-zero shear. See *Specified Shear* (p. 304) for details. This option is not available for moving walls.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Marangoni Stress
enables the specification of shear stress caused by the variation of surface tension due to temperature. This option is not available for moving walls.

Shear Stress
contains inputs related to wall shear. These items will appear when **Specified Shear** is selected as the **Shear Condition**. See *Specified Shear* (p. 304) for details.

X-Component, Y-Component, Z-Component, Swirl Component
specify the x, y, and z or swirl components of shear for a slip wall. **Swirl Component** is available only for axisymmetric swirl cases.

Specularity Coefficient
is used in multiphase granular flow. You can specify the specularity coefficient such that when the value is zero, this condition is equivalent to zero shear at the wall, but when the value is near unity, there is a significant amount of lateral momentum transfer.

Specularity Coefficient
allows you to enter a value between zero and one, which controls the amount of lateral momentum transfer.

Marangoni Stress
contains inputs related to Marangoni stress. This item will appear when **Marangoni Stress** is selected as the **Shear Condition**. See *Marangoni Stress* (p. 305) for details.

Surface Tension Gradient
specifies the surface tension gradient with respect to temperature ($d\sigma/dT$ in *Equation 7–82* (p. 305)).

Wall Roughness
contains inputs for defining wall roughness in turbulent calculations. See *Wall Roughness Effects in Turbulent Wall-Bounded Flows* (p. 306) for details.

Roughness Height
sets the roughness height K_s (see *Setting the Roughness Parameters* (p. 308) for details).

Roughness Constant
sets the roughness constant C_{K_s} (see *Setting the Roughness Parameters* (p. 308) for details).

Wall Adhesion
contains inputs related to wall adhesion. This section of the dialog box will appear if you are using the VOF model and have enabled wall adhesion in the e *Phase Interaction Dialog Box* (p. 1791).

Contact Angles
specifies the contact angle at the wall for each pair of phases (θ_w in *Figure 26.25* (p. 1141)in the Theory Guide). See *Steps for Setting Boundary Conditions* (p. 1115) for details.

Thermal
contains the thermal parameters. This tab is available only when the energy equation is turned on.

Thermal Conditions
contains radio buttons for selecting the thermal boundary condition type. See *Thermal Boundary Conditions at Walls* (p. 309) for details about these inputs:

Heat Flux
selects a specified heat flux condition.

Temperature
selects a specified wall temperature condition.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Convection
selects a convective heat transfer boundary condition model.

Radiation
selects an external radiation boundary condition.

Mixed
selects a combined convection/external radiation boundary condition.

Coupled
selects a coupled heat transfer condition. It is applicable only to walls that form the interface between two regions (such as the fluid/solid interface for a conjugate heat transfer problem).

Once a condition type has been selected, the appropriate conditions can be specified.

Heat Flux
sets the wall heat flux to be used for the **Heat Flux** condition. A specification of zero **Heat Flux** is simply the adiabatic condition (no heat transfer). A positive value of heat flux implies that heat is input *into* the domain.

Temperature
sets the wall temperature to be used for the **Temperature** condition.

Heat Transfer Coefficient
sets the convective heat transfer coefficient to be used for the **Convection** condition (h_{eff} in *Equation 7–94* (p. 318)).

Free Stream Temperature
sets the reference or free stream temperature to be used for the **Convection** condition (T_{ext} in *Equation 7–94* (p. 318)).

External Emissivity
sets the emissivity of the external wall to be used for the **Radiation** condition (ε_{ext} in *Equation 7–95* (p. 318)).

External Radiation Temperature
sets the temperature of the external radiation source/sink to be used for the **Radiation** condition (T_{∞} in *Equation 7–95* (p. 318)).

Internal Emissivity
sets the internal emissivity of the wall. This item will appear only if you are using the gray P-1 model, the DTRM, the gray discrete ordinates model, or the S2S model for radiation heat transfer. (Note that it will not appear if you are using the non-gray P-1 model or the non-gray discrete ordinates model. In these cases, you will enter the **Internal Emissivity** for each band in the **Radiation** tab.)

Wall Thickness
sets the thickness of the wall for calculation of thin-wall thermal resistance. (See *Thin-Wall Thermal Resistance Parameters* (p. 311) for details.)

Heat Generation Rate
sets the rate of heat generation in the wall.

Contact Resistance
sets the contact resistance (R_c in Equation 18–23 in the Theory Guide) at the wall. See *Modeling Solidification and Melting* (p. 1201) for details. This item appears only when the solidification/melting model is used.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Name

sets the material type of the wall. The conductivity of the material is used for the calculation of thin-wall thermal resistance. (See *Thin-Wall Thermal Resistance Parameters* (p. 311) for details.) **Material** is used only when **Wall Thickness** is non-zero. Materials are defined with the *Materials Task Page* (p. 1735).

Shell Conduction

enables shell conduction for the wall. See *Shell Conduction in Thin-Walls* (p. 313) for details.

Radiation

displays the boundary conditions for the S2S model, the DO model, and the non-gray P-1 model at the wall. This tab is only available if you are using the surface to surface model, the discrete ordinates model, or the non-gray P-1 model. See *Forming Surface Clusters* (p. 711), *Wall Boundary Conditions for the DO Model* (p. 726), and *Wall Boundary Conditions for the DTRM, and the P-1, S2S, and Rosseland Models* (p. 725) for details.

BC Type

contains a drop-down list of available radiation boundary condition types. The available options are **opaque** and **semi-transparent**. This item will appear only if you are using the discrete ordinates model.

Internal Emissivity

specifies the internal emissivity of the wall in each wavelength band. This item will appear only if you are using the non-gray discrete ordinates model and you have selected **opaque** as the **BC Type**, or if you are using the non-gray P-1 model.

Diffuse Fraction

specifies the fraction of the irradiation that is to be treated as diffuse. By default, the **Diffuse Fraction** is set to 1, indicating that all of the irradiation is diffuse. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band. This item will appear only if you are using the discrete ordinates model.

Beam Width

specifies the beam width for an external semi-transparent wall in terms of the **Theta** and **Phi** extents. This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Beam Direction

specifies the beam direction as an **X,Y,Z** vector. You can specify the **Beam Direction** as a constant, a profile, or a UDF. This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Direct Irradiation

specifies the value of the irradiation flux. If the non-gray DO model is being used, a constant **Direct Irradiation** can be specified for each band.

This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Apply Direct Irradiation Parallel to the Beam

is the default means of specifying the scale of irradiation flux. When enabled, ANSYS FLUENT assumes that the value of **Direct Irradiation** that you specify is the irradiation flux parallel to the **Beam Direction**. When deselected, ANSYS FLUENT instead assumes that the value specified is the flux parallel to the face normals and will calculate the resulting beam parallel flux for every face. This item will appear only if you are using the discrete ordinates model.

Diffuse Irradiation

specifies the value of the irradiation flux. If the non-gray DO model is being used, a constant **Diffuse Irradiation** can be specified for each band.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Solar Boundary Conditions

contains the settings for solar ray tracing. This group box is available only if you select **Solar Ray Tracing** from the **Model** list in the **Solar Load** group box of the **Radiation Model** dialog box. See *Solar Ray Tracing* (p. 754) for details.

Participates in Solar Ray Tracing
specifies whether or not the wall participates in solar ray tracing.

Absorptivity
contains the settings that define the absorptivity of wall.

Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the absorption of the wall. The value should be defined for normal incident rays.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the absorption of the wall. The value should be defined for normal incident rays.

Diffuse Hemispherical
specifies a multiplier (ranging from 0 to 1) that is applied to the diffuse solar radiation to account for the absorption of the wall. This setting is only available for semi-transparent walls.

Transmissivity
contains the settings that define the transmissivity of wall. This group box is only available when **semi-transparent** is selected for **BC Type**.

Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the transmissivity of the wall. The value should be defined for normal incident rays. This setting is only available for semi-transparent walls.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the transmissivity of the wall. The value should be defined for normal incident rays. This setting is only available for semi-transparent walls.

Diffuse Hemispherical
specifies a multiplier (ranging from 0 to 1) that is applied to the diffuse solar irradiation to account for the transmissivity of the wall. This setting is only available for semi-transparent walls.

S2S Parameters
contains the settings for the S2S radiation model. This group box is available only if you select the **Surface to Surface** radiation model. See *Forming Surface Clusters* (p. 711) for details.

Faces Per Surface Cluster
sets the number of faces per surface cluster (FPSC) for the wall, and thus controls the number of radiating surfaces and (if you select **Cluster to Cluster** for **Basis** in the **View Factors and Clustering** dialog box) view factor surfaces.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Critical Zone
specifies that the wall is a critical zone. When this option is enabled, the value entered for **Faces Per Surface Cluster** will not be altered when you use **Automatic** clustering in the **View Factors and Clustering** dialog box, and impacts the calculations and actions performed by the buttons in the **Maximum Distance from Critical Zone** group box of the **Participating Boundary Zones** dialog box.

Participates in View Factor Calculation
specifies whether or not the wall participates in the view factor calculation as part of the S2S radiation model. This option is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters. This tab is available only if you have enabled the **Species Transport** model in the *Species Model Dialog Box* (p. 1681).

Reaction
activates reactions at the wall. This item will appear only if you have enabled any of the reactions in the *Species Model Dialog Box* (p. 1681).

Reaction Mechanisms
allows you to specify a defined group, or mechanism, of available reactions. This item will appear only if the **Reaction** option has been turned on. See *Defining Zone-Based Reaction Mechanisms* (p. 816) for details about defining reaction mechanisms.

Surface Area Washcoat Factor
allows you to specify a factor, which multiplies the wall area to account for the increased surface area of washcoats. This item will appear only if the **Reaction** option has been turned on. See *Species Boundary Conditions for Walls* (p. 314) for details.

Species Boundary Condition
contains options for the specification of species boundary conditions. See *Species Boundary Conditions for Walls* (p. 314) for details.

Zero Diffusive Flux
indicates a zero-flux condition for a species. This is the default condition.

Specified Mass Fraction
indicates that the species mass fraction will be specified.

Species Mass Fractions
contains inputs for the species mass fractions of any species for which you have selected **Species Mass Fraction** as the **Species Boundary Condition**.

DPM
contains the discrete phase parameters. This tab is available only if you have defined at least one injection.

Discrete Phase Model Conditions
contains inputs for setting the fate of particle trajectories at the wall. These options will appear when one or more injections have been defined. See *Setting Boundary Conditions for the Discrete Phase* (p. 1050) for details.

Boundary Cond. Type
sets the way that the discrete phase behaves with respect to the boundary.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficients of restitution. (See *Figure 25.19* (p. 1051))

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See *Figure 25.20* (p. 1051).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See *Figure 25.21* (p. 1052).

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Boundary Cond. Function
sets the user-defined function from the drop-down list.

Discrete Phase Reflection Coefficients
determine the behavior of reflecting particles. This item appears when **reflect** is chosen as the **Boundary Cond. Type**. See *Discrete Phase Boundary Condition Types* (p. 1052) for details on setting the following items.

Normal
sets the type of function for the normal coefficient of restitution. This function can be **constant**, **piecewise-linear**, **piecewise-polynomial**, or **polynomial**.

Tangent
sets the type of function for the tangential coefficient of restitution. This function can be **constant**, **piecewise-linear**, **piecewise-polynomial**, or **polynomial**.

Erosion Model
contains inputs for erosion calculations. See *Discrete Phase Boundary Condition Types* (p. 1052) for details about these items.

Impact Angle Function
specifies the value of $f(\alpha)$ in Equation 16–205 in the Theory Guide.

Diameter Function
specifies the value of $C\left(d_p\right)$ in Equation 16–205 in the Theory Guide.

Velocity Exponent Function
specifies the value of $b(v)$ in Equation 16–205 in the Theory Guide.

UDS
displays the boundary conditions for user-defined scalars (UDSs) at the wall. This tab is available only if you have specified a non-zero number of user-defined scalars in the *User-Defined Scalars Dialog Box* (p. 2113).

User Defined Scalar Boundary Condition
contains options for the specification of UDS boundary conditions. See the separate UDF Manual for details.

Specified Flux
indicates that the flux of the UDS at the wall will be specified.

Specified Value
indicates that the value for the UDS at the wall will be specified.

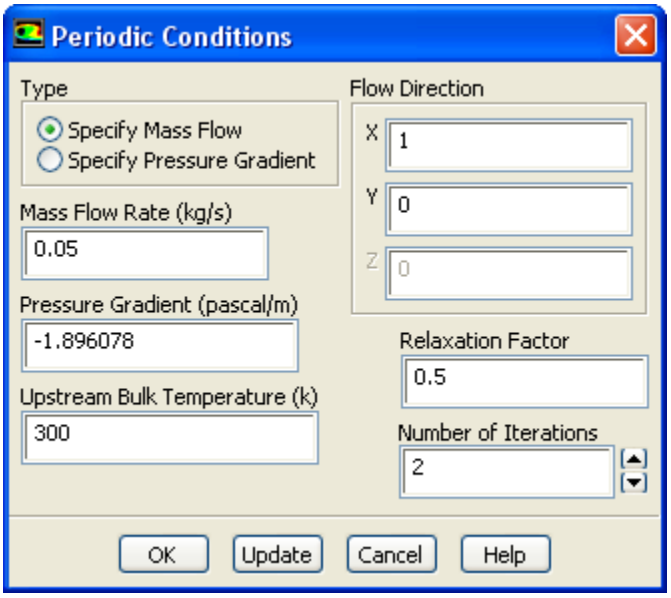
User Defined Scalar Boundary Value
contains inputs for the value of the flux of the UDS, or the value of the UDS itself, depending on your selection for that UDS under **User Defined Scalar Boundary Condition**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.7.21. Periodic Conditions Dialog Box

The **Periodic Conditions** dialog box allows you to set parameters that define fully-developed periodic flow and heat transfer. See *User Inputs for the Pressure-Based Solver* (p. 494) and *Using Periodic Heat Transfer* (p. 765) for details.

(This dialog box is available only when the pressure-based solver is used; it is not available for the density-based coupled solvers.)



Controls

Specify Mass Flow
enables the specification of the mass flow rate.

Specify Pressure Gradient
enables the specification of the pressure gradient.

Mass Flow Rate
specifies the mass flow rate. This item will not be available if you selected the **Specify Pressure Gradient** option.

Important

For axisymmetric problems, the mass flow rate is per 2π radians.

Pressure Gradient
specifies the pressure gradient (β in Equation 1–22 in the Theory Guide).

Upstream Bulk Temperature
sets the inlet bulk temperature for periodic heat transfer calculations.

Flow Direction
sets the direction of the periodic flow. The direction vector must be parallel to the periodic translation direction or its opposite.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Relaxation Factor

sets the under-relaxation factor that controls convergence of the iteration process described in *Setting Parameters for the Calculation of β* (p. 496) for specified mass flow.

Number of Iterations

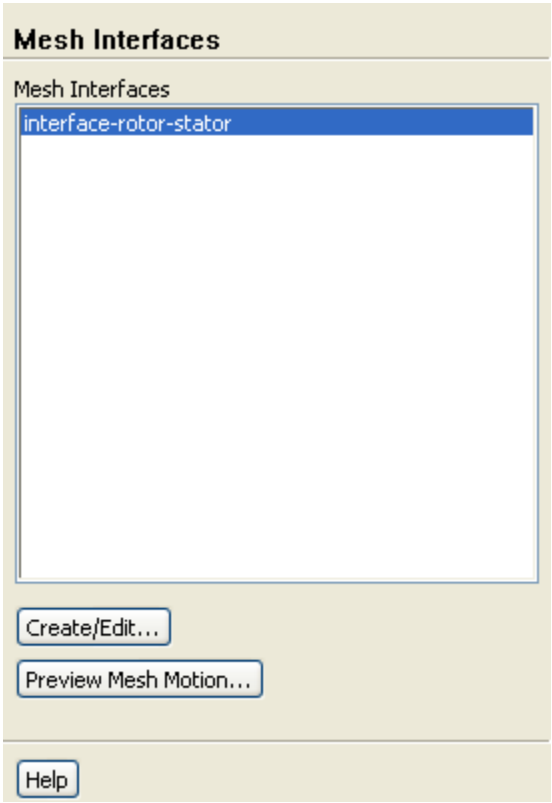
sets the number of subiterations done on the correction of β in the pressure correction equation for specified mass flow. See *Setting Parameters for the Calculation of β* (p. 496) for details.

Update

updates the **Pressure Gradient** field with the current value.

35.8. Mesh Interfaces Task Page

The **Mesh Interfaces** task page allows you to define the parameters for any mesh interfaces in your model.



Controls

Mesh Interfaces

contains a list of mesh interfaces.

Create/Edit...

displays the *Create/Edit Mesh Interfaces Dialog Box* (p. 1876).

Preview Mesh Motion...

displays the *Mesh Motion Dialog Box* (p. 1896).

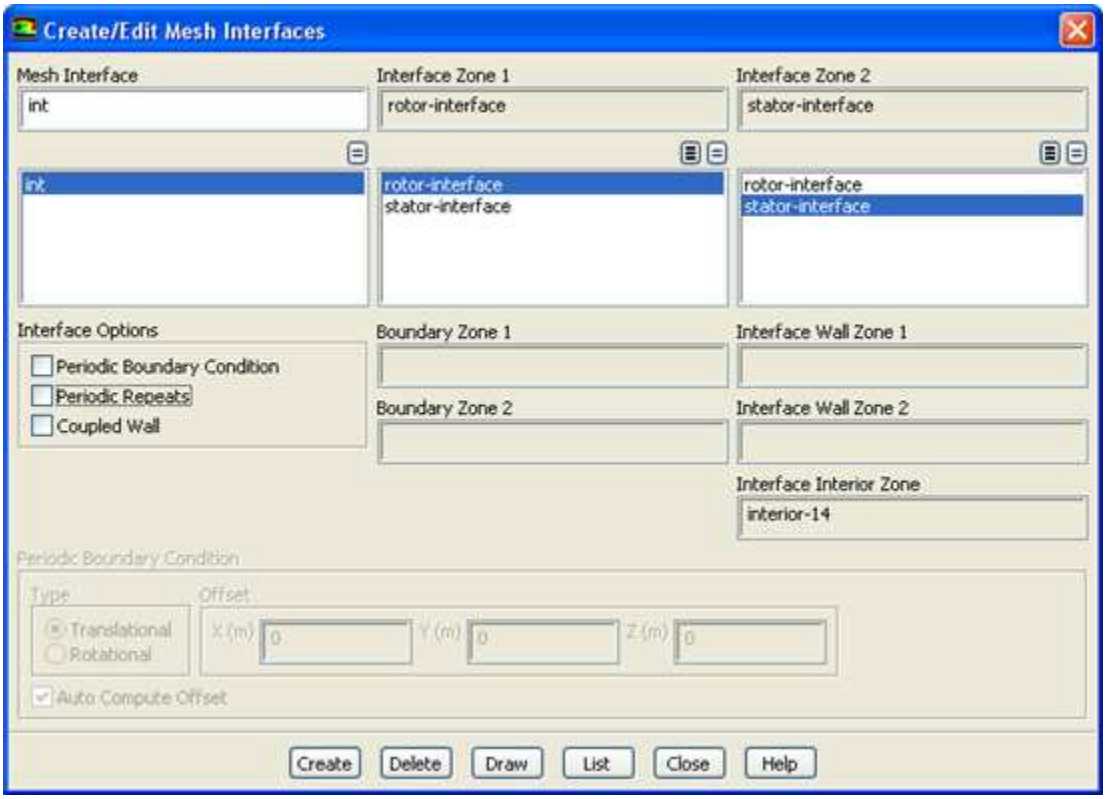
For additional information, please see the following section:

35.8.1. Create/Edit Mesh Interfaces Dialog Box

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.8.1. Create/Edit Mesh Interfaces Dialog Box

The **Create/Edit Mesh Interfaces** dialog box allows you to define mesh interfaces for use with sliding meshes (see *Using Sliding Meshes* (p. 540)) or multiple reference frames (see *Mesh Setup for a Multiple Moving Reference Frame* (p. 524)), or for meshes with non-conformal boundaries (see *Non-Conformal Meshes* (p. 161)).



Controls

Mesh Interface

contains a text entry box in which you can set the name of the mesh interface, and a list from which you can select an existing mesh interface.

Interface Zone 1, Interface Zone 2

contain lists from which you can select the two interface zones that comprise the mesh interface, and informational fields that show the name of the zone you selected in each list. (You cannot edit these fields; the name in this field will be the name of the zone you selected in the list below it.)

Interface Options

contains options related to the interface type.

Periodic Boundary Condition

allows you to create a non-conformal periodic boundary condition interface.

Periodic Repeats

is relevant when you have two zones coming together, such as a rotor and stator. When the two zones move, a portion of the geometry will intersect and that will be an interior zone, but on either side of the interior, the zone is termed periodic repeats.

Coupled Wall

indicates (if enabled) that the interface acts as a thermally coupled wall.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Zone 1, Boundary Zone 2
display the names of the wall boundary zones that ANSYS FLUENT creates during the creation of a non-periodic mesh interface. If the two interface zones overlap each other, then the wall boundaries are created but with zero faces.

Interface Interior Zone
displays the names of the interface interior zones created. This is used for embedded LES, when there is a need to be able to convert an interior zone into a RANS-LES interface.

Interface Wall Zone 1, Interface Wall Zone 2
display the names of the wall interface zones (e.g., `wall-4`, `wall-4-shadow`), which are created if the **Coupled Wall** option is enabled.

Periodic Boundary Condition

Type
allows you to select a periodicity that is either **Translational** or **Rotational**.

Offset
is the offset coordinates or angle, depending on whether **Translational** or **Rotational** periodicity is selected. Note that when **Auto Compute Offset** is enabled, the **Offset** fields are not editable.

Auto Compute Offset
will result in ANSYS FLUENT finding the offset. If this option is disabled, then you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Create
creates the specified mesh interface (and gives it the name specified under **Mesh Interface**).

Delete
deletes the mesh interface selected under **Mesh Interface**.

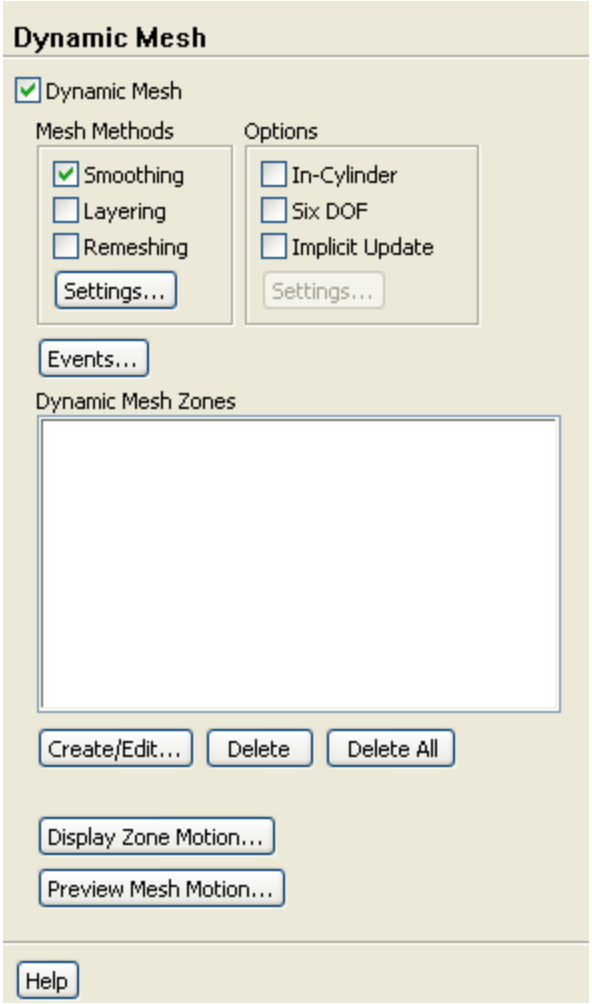
Draw
allows you to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display interface zones from **Interface Zone 1** or Interface Zone 2 prior to defining any **Mesh Interfaces**. After a **Mesh Interface** is defined, you can select the appropriate mesh interface and click the **Draw** button to display the zones under **Interface Zone 1** and Interface Zone 2 together as defined by the **Mesh Interface**.

List
lists information about the selected **Mesh Interface**. When you click on this button, ANSYS FLUENT will report (in the console window) the two interface boundaries and all new zones that were created (i.e., interior, wall, or periodic zones).

35.9. Dynamic Mesh Task Page

The **Dynamic Mesh** task page allows you to define the all the parameters for modeling a dynamic mesh model. See *Setting Dynamic Mesh Modeling Parameters* (p. 550) for details about using the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Dynamic Mesh

enables the dynamic mesh model and activates the controls in the task page.

Mesh Methods

contains options to specify the mesh update method(s).

Smoothing

enables the spring-based or diffusion-based smoothing methods. In spring-based smoothing, edges between any two mesh nodes are idealized as a network of interconnected springs. In diffusion-based smoothing, the interior mesh motion is governed by the solution to a diffusion problem. See *Spring-Based Smoothing* (p. 552) for details.

Layering

enables the dynamic layering method which can be used to add or remove layers of cells adjacent to a moving boundary based on the height of the layer adjacent to the moving surface in prismatic mesh zones. See *Dynamic Layering Method* (p. 564) for details.

Remeshing

enables the local or zonal remeshing methods. In local remeshing, the cells that violate the skewness or size criteria are agglomerated and locally remeshed; see *Local Remeshing Method* (p. 570) for further details. In zonal remeshing, the complete cell zone (including the boundary zones) is remeshed; see *CutCell Zone Remeshing Method* (p. 579) for further details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Settings...
displays the *Mesh Method Settings Dialog Box* (p. 1879).

Options
contains options to specify specialized dynamic mesh models.

In-Cylinder
enables the in-cylinder model. See *In-Cylinder Settings* (p. 587) for more information.

Six DOF
enables six degrees of freedom solver. See *Using the Six DOF Solver* (p. 602) for more information.

Implicit Update
specifies that the mesh is updated during a time step (as opposed to just at the beginning of a time step). See *Implicit Update Settings* (p. 603) for more information.

Settings...
opens the *Options Dialog Box* (p. 1883), where you can set the parameters for the options that are enabled in the **Options** group box.

Events...
displays the *Dynamic Mesh Events Dialog Box* (p. 1886).

Dynamic Mesh Zones
displays a list of dynamic mesh zones.

Create/Edit...
displays the *Dynamic Mesh Zones Dialog Box* (p. 1889).

Delete
removes the selected dynamic zone(s) from the **Dynamic Mesh Zones** list.

Delete All
removes all dynamic zones from the **Dynamic Mesh Zones** list.

Display Zone Motion...
displays the *Zone Motion Dialog Box* (p. 1895).

Preview Mesh Motion...
displays the *Mesh Motion Dialog Box* (p. 1896).

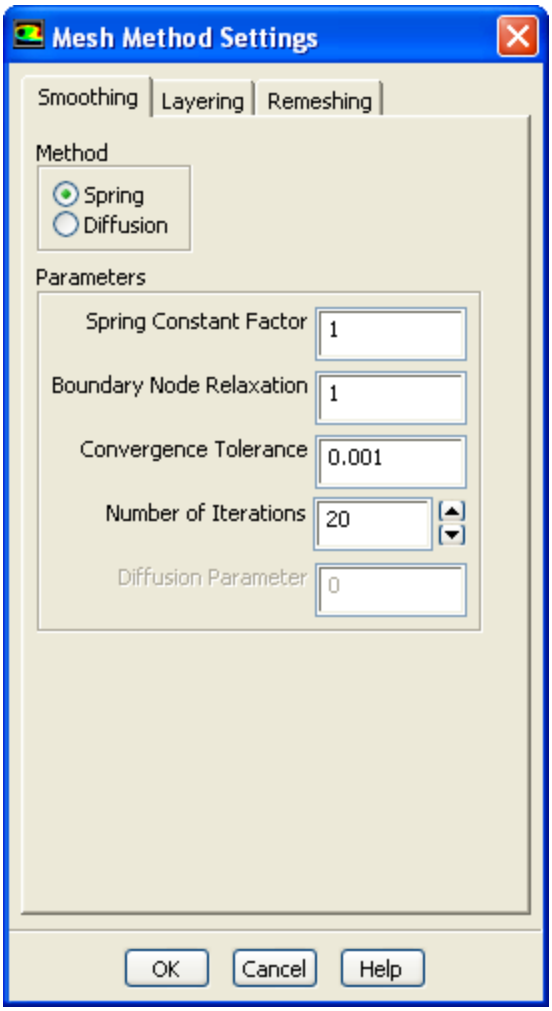
For additional information, please see the following sections:

35.9.1. Mesh Method Settings Dialog Box
35.9.2. Mesh Scale Info Dialog Box
35.9.3. Options Dialog Box
35.9.4. In-Cylinder Output Controls Dialog Box
35.9.5. Dynamic Mesh Events Dialog Box
35.9.6. Define Event Dialog Box
35.9.7. Events Preview Dialog Box
35.9.8. Dynamic Mesh Zones Dialog Box
35.9.9. CutCell Boundary Zones Info Dialog Box
35.9.10. Zone Scale Info Dialog Box
35.9.11. Zone Motion Dialog Box
35.9.12. Mesh Motion Dialog Box
35.9.13. Autosave Case During Mesh Motion Preview Dialog Box

35.9.1. Mesh Method Settings Dialog Box

The **Mesh Method Settings** dialog box allows you to apply settings for the smoothing, layering, or remeshing methods.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Smoothing

contains parameters to be specified for the smoothing mesh update method.

Method

gives you a choice of using spring-based or diffusion-based smoothing.

Spring

selects the spring-based smoothing method.

Diffusion

selects the diffusion-based smoothing method.

Spring Constant Factor

controls the spring stiffness.

Boundary Node Relaxation

specifies how the node positions on the deforming boundaries are updated. This applies only if your model contains deforming boundaries.

Convergence Tolerance

controls the smoothing convergence.

Number of Iterations

specifies the number of iterations.

Diffusion Parameter

specifies α.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layering

contains parameters to be specified for the layering mesh update method.

Options

specifies the criteria for splitting or collapsing cell layers.

Height Based

specifies that the cell layers are split or merged based on height.

Ratio Based

specifies that the cell layers are split or merged based on ratios.

Split Factor

specifies the value of α_s in*Equation 11–14* (p. 565). It controls the height or ratio at which the cells are split.

Collapse Factor

specifies the value of α_c in *Equation 11–15* (p. 566). It controls the height or ratio at which the cells are collapsed and merged into the next layer.

Remeshing

contains parameters to be specified for the remeshing mesh update method.

Remeshing Methods

contain options that control remeshing.

Local Cell

allows you to remesh deforming boundary cells.

Local Face

allows you to remesh deforming boundary faces. This option is available for 3D cases.

Region Face

allows you to remesh a region.

CutCell Zone

allows you to replace an entire cell zone with a predominantly Cartesian mesh. This option is only available for 3D cases. Note that the parameters that control this remeshing method are set in the **Dynamic Mesh Zones** dialog box. See *Using the CutCell Zone Remeshing Method* (p. 581) for more information.

2.5D

enables the 2.5D model. This option is only available for 3D cases. See *Using the 2.5D Model* (p. 583) for more information.

Parameters

contains parameters that control remeshing for all of the remeshing methods except for **CutCell Zone**.

Minimum Length Scale

specifies the lower limit of cell size below which the cells are marked for remeshing.

Maximum Length Scale

specifies the upper limit of cell size above which the cells are marked for remeshing.

Maximum Cell Skewness

specifies the desired maximum skewness for the mesh.

Maximum Face Skewness

specifies the desired maximum skewness for the surface mesh. This option is active, when **Local Face** is selected under **Remeshing Methods**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Size Remeshing Interval
specifies the interval in time steps for remeshing based on the above size criteria only. Marking of cells based on skewness occurs automatically at every time step when **Remeshing** is enabled.

Mesh Scale Info...
opens the *Mesh Scale Info Dialog Box* (p. 1882), in which you can view the statistics of the mesh, such as the minimum and maximum length scale values and the maximum cell and face skewness values.

Use Defaults
resets the remeshing parameters to the default values.

Sizing Function
contains parameters that control the sizing function.

On
allows you to enable or disable the sizing function.

Resolution
sets the resolution for the sizing function. See *Setting Dynamic Mesh Modeling Parameters* (p. 550) for more information. This item will appear only if **Sizing Function** is enabled.

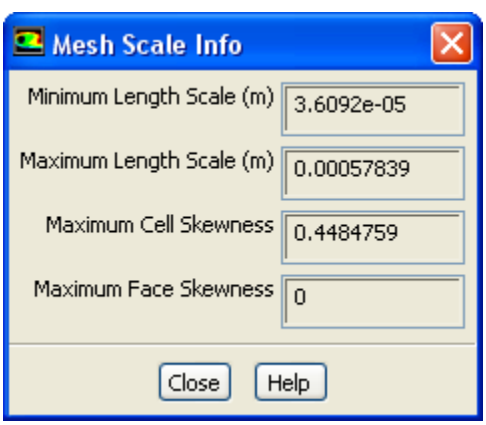
Variation
specifies the value of α in *Equation 11–21* (p. 575). This item will appear only if **Sizing Function** is enabled.

Rate
specifies the value of β in *Equation 11–22* (p. 575). This item will appear only if **Sizing Function** is enabled.

Use Defaults
resets the sizing function parameters to the default values. This item will appear only if **Sizing Function** is enabled.

35.9.2. Mesh Scale Info Dialog Box

The **Mesh Scale Info** dialog box allows you to inspect the values of minimum and maximum length scale and maximum cell/face skewness in a mesh.



Controls

Minimum Length Scale
displays the lower limit of cell size below which the cells are marked for remeshing.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

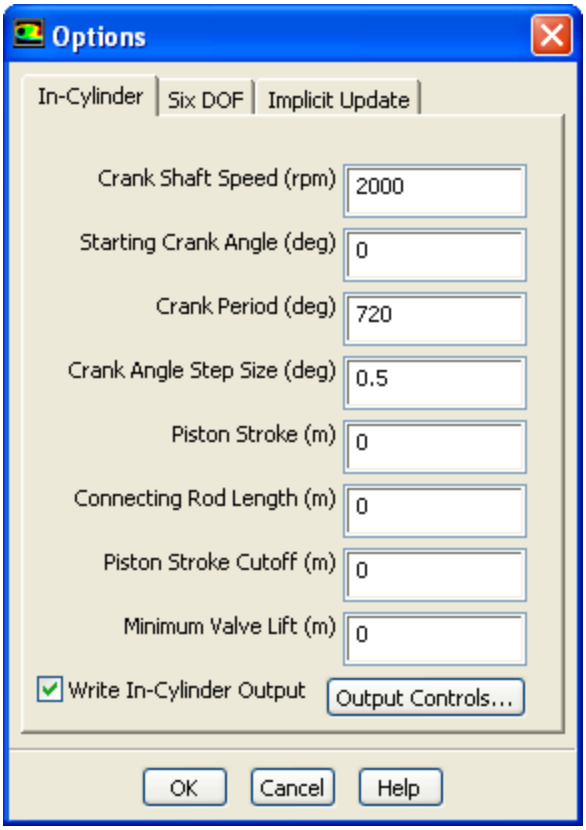
Maximum Length Scale
displays the upper limit of cell size below which the cells are marked for remeshing.

Maximum Cell Skewness
displays the maximum cell skewness in the zone.

Maximum Face Skewness
displays the maximum cell skewness in the surface mesh.

35.9.3. Options Dialog Box

The **Options** dialog box allows you to set the parameters for the options available in the **Options** group box of the **Dynamic Mesh** task page.



In-Cylinder
contains parameters to be specified for the in-cylinder model. See *In-Cylinder Settings* (p. 587) for more information.

Crank Shaft Speed
specifies the speed of the crank shaft.

Starting Crank Angle
specifies the starting crank angle.

Crank Period
specifies the crank period.

Crank Angle Step Size
specifies the crank angle step size used to determine the time step size to advance the solution.

Piston Stroke
specifies the piston stroke to calculate the piston location.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Connecting Rod Length
specifies the length of the connecting rod.

Piston Stroke Cutoff
specifies the piston stroke cutoff used to control the onset of layering in the cylinder chamber.

Minimum Valve Lift
specifies the minimum valve lift.

Write In-Cylinder Output
enables or disables the writing of in-cylinder specific output parameters. When this option is enabled, the **Output Controls...** button becomes active.

Output Controls...
displays the *In-Cylinder Output Controls Dialog Box* (p. 1884) and is available only after the **Write In-Cylinder Output** option is enabled.

Six DOF
contains parameters to be specified for the six DOF solver. See *Six DOF Solver Settings* (p. 601) for more information.

Gravitational Acceleration
contains the text entry boxes for gravitational acceleration in X, Y, and Z directions.

X, Y, Z
specifies the gravitational acceleration in X, Y, and Z directions respectively.

Write Motion History
allows you to keep track of an object's motion history.

File Name
allows you to specify a file name for saving the object's motion history.

Implicit Update
contains parameters to be specified for implicit mesh updating. See *Implicit Update Settings* (p. 603) for more information.

Update Interval
allows you to specify the frequency in iterations at which the mesh will be updated within a time step.

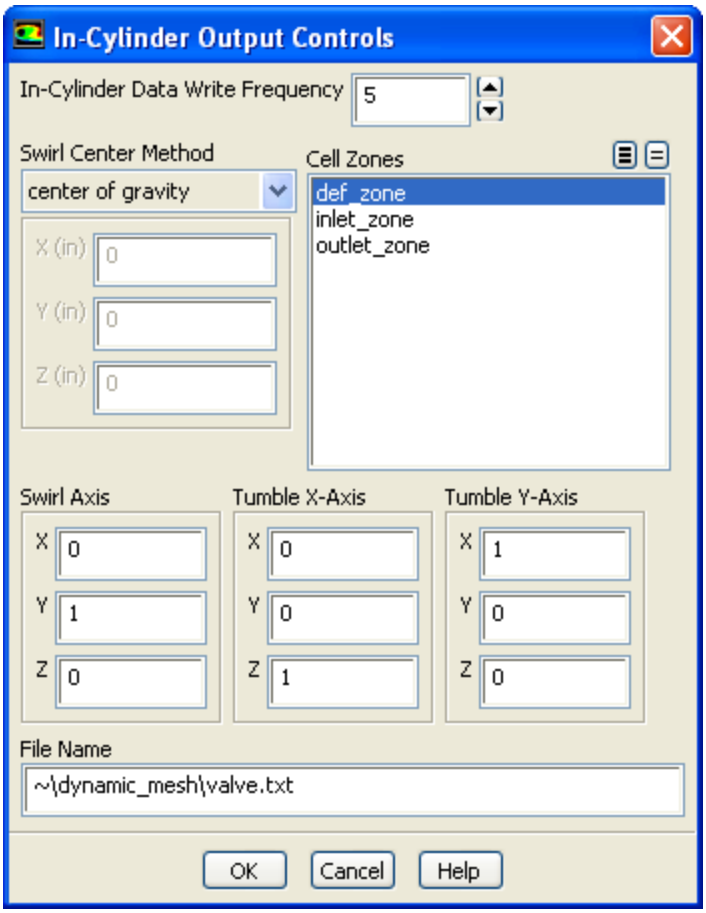
Motion Relaxation
allows you to set a value (within the range of 0 to 1) for ω in *Equation 11–28* (p. 604), which defines the relaxation of the motion (i.e., displacement of the nodes) during the mesh update.

Residual Criteria
allows you to set the relative residual threshold that is used to check the motion convergence.

35.9.4. In-Cylinder Output Controls Dialog Box

The **In-Cylinder Output Controls** dialog box contains parameters that control the output for the in-cylinder model. See *In-Cylinder Settings* (p. 587) for more information.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

In-Cylinder Data Write Frequency

represents an integer entry specifying the interval in number of time-steps. Make sure that a value other than 0 is used for the frequency, in order to allow you to complete your setup.

Swirl Center Method

contains a drop-down list which allows you to select the method to calculate the swirl center. The list contains **center of gravity** and **fixed**, with **center of gravity** being the default value.

center of gravity

calculates the swirl center inside the code and is used as the center of gravity of the chosen cell zones.

fixed

enables you to specify a swirl center in the **X**, **Y**, and **Z** entry fields below the drop-down list.

In addition to these two options, you can chose to use your own **Compiled** UDF to calculate the swirl center. For details on using a dynamic mesh UDF, see the separate for information on user-defined functions.

Cell Zones

is a list which displays the names of all existing cell zones in the case files. You can select only the zones relevant for the swirl and tumble calculations.

Swirl Axis

specifies the swirl axis with three entries for the directional components. By default, **X**, **Y**, **Z**= **0**, **1**, **0**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tumble X-Axis

specifies the directional components of **Tumble X-Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **0**, **0**, **1**. This applies only in 3D.

Tumble Y-Axis

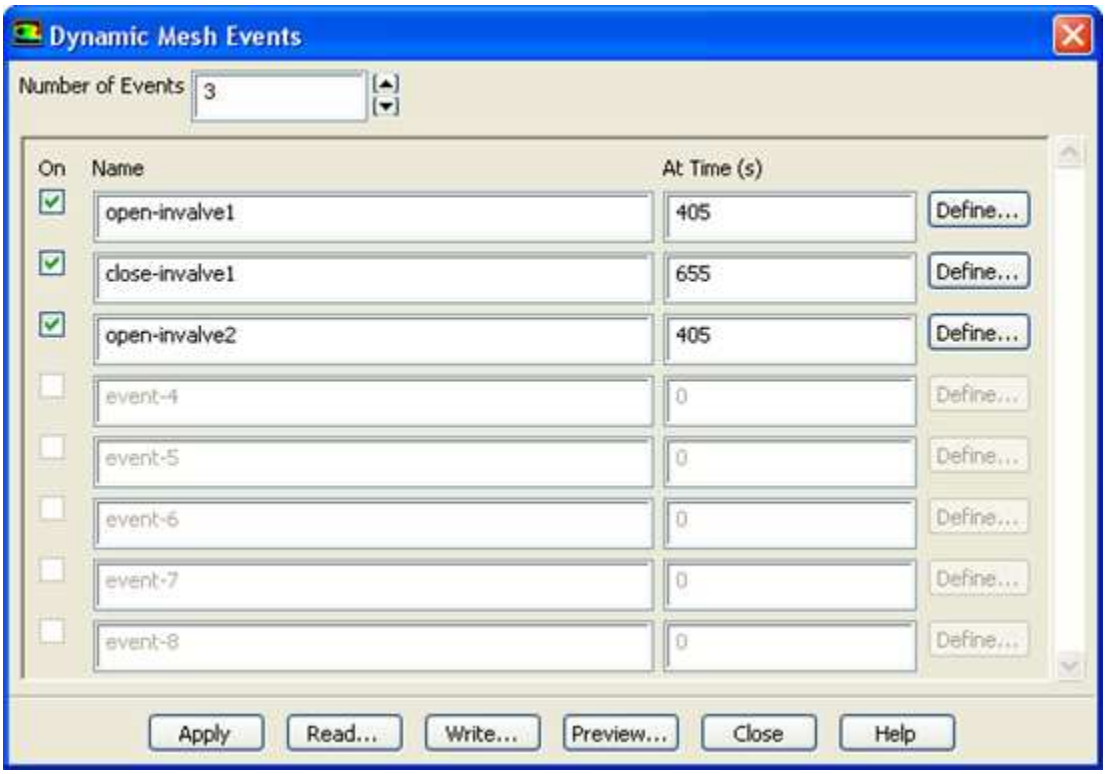
specifies the directional components of **Tumble Y-Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **1**, **0**, **0**. This applies only in 3D.

File Name

specifies the name of the **In-Cylinder** output file. By default, the file name contains the name of the case file appended with a `.txt` extension.

35.9.5. Dynamic Mesh Events Dialog Box

The **Dynamic Mesh Events** dialog box is available to control the timing of specific events during the course of the simulation. See *Defining Dynamic Mesh Events* (p. 605) for details.



Controls

Number of Events

specifies the number of events to be defined.

On

enables the corresponding event.

Name

specifies the name of the event.

At Crank Angle

specifies the angular location of the crank at which the event should occur. This option appears for in-cylinder flows.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At Time
specifies the time (in seconds) at which you want the event to occur. This option appears for non-in-cylinder flows.

Define...
opens the *Define Event Dialog Box* (p. 1887).

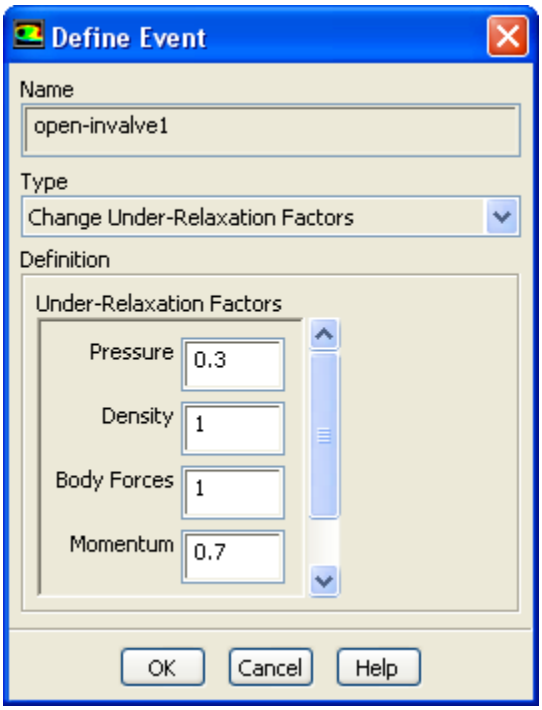
Read...
opens *The **Select File** Dialog Box* (p. 33).

Write...
opens *The **Select File** Dialog Box* (p. 33).

Preview...
opens the *Events Preview Dialog Box* (p. 1889).

35.9.6. Define Event Dialog Box

The **Define Event** dialog box allows you to define events.



Controls

Name
contains the name of the event to be defined.

Type
specifies the type of event. You can choose the type of event from the drop-down list. These event types and their definitions are described in *Events* (p. 608).

Definition
contains the input parameters corresponding to the type of event selected under **Type**.

Zone
contains a selectable list of the zones. The selection specifies the name of the zone(s) to be changed. This item will appear only for the **Change Zone Type** event.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Zone Type
specifies the type of zone that an existing zone needs to be changed to. This item will appear only for the **Change Zone Type** event.

From Zone
specifies the name of the zone from which the boundary condition is to be copied. This item will appear only for the **Copy Zone BC** event.

To Zone(s)
specifies the name of the zone to which the boundary condition is to be copied. This item will appear only for the **Copy Zone BC** event.

Zone(s)
contains a selectable list of the zones. The selection specifies the name of the zone to be activated / deactivated. This item will appear only for the **Activate Cell Zone** and **Deactivate Cell Zone** events.

Interface Name
contains the name of the interface to be created or deleted. This item will appear only for the **Create Sliding Interface** and **Delete Sliding Interface** events.

Interface Zone 1, Interface Zone 2
specifies the two zones on either side of the interface to be created. This item will appear only for the **Create Sliding Interface** event.

Wall 1 Motion, Wall 2 Motion
specifies the dynamic zones whose motion can be copied for the zones specified under **Interface Zone 1** and **Interface Zone 2**. This item will appear only for the **Create Sliding Interface** event.

Attribute
allows you to select the relevant motion attribute. This item will appear only for the **Change Motion Attribute** event.

Status
allows you to enable or disable the motion attribute. This item will appear only for the **Change Motion Attribute** event.

Dynamic Mesh Zones
contains a list of dynamic zones. This item will appear only for the **Change Motion Attribute** event.

Crank Angle Step Size
specifies the new physical time step value in degrees. This item will appear only for the **Change Time Step Size** event.

Base Dynamic Zone
specifies the zone from which the layer of cells is to be created. This item will appear only for the **Insert Boundary Layer**and **Remove Boundary Layer** event.

Side Dynamic Zone
represents the deforming face zone adjacent to the **Base Dynamic Zone** before the layer is inserted. This item will appear only for the **Insert Boundary Layer** event.

Internal Zone 1 Name, Internal Zone 2 Name
specifies the name of the new internal zones.

Time Step Size
allows you to change the time step size of the event. This item will appear only for the **Change Time Step Size** event.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under-Relaxation Factors
allows you to specify the under-relaxation factors for the selected event. This item will appear only for the **Change Under-Relaxation Factors** event. See *Setting Under-Relaxation Factors* (p. 1217) for details.

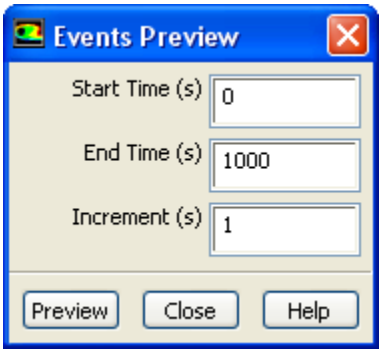
Adjacent Dynamic Face Zone
allows you to select the dynamic face zone adjacent to the location of the cell layer to be inserted (or deleted). You can select the required zone from the drop-down list.

Direction Parameter
is the direction with respect to the selected dynamic face zone a layer of cells is removed or added.

Command
specifies the series of text / Scheme commands or the macro to be executed during the simulation. This item will appear only for the **Execute Command** event.

35.9.7. Events Preview Dialog Box

The **Events Preview** dialog box allows you to play the events to check that they are defined correctly.



Controls

Start Crank Angle
specifies the crank angle at which you want to start the preview (for in-cylinder flows).

Start Time
specifies the time at which you want to start the preview (for non-in-cylinder flows).

End Crank Angle
specifies the crank angle at which you want to end the preview (for in-cylinder flows).

End Time
specifies the time at which you want to end the preview (for non-in-cylinder flows).

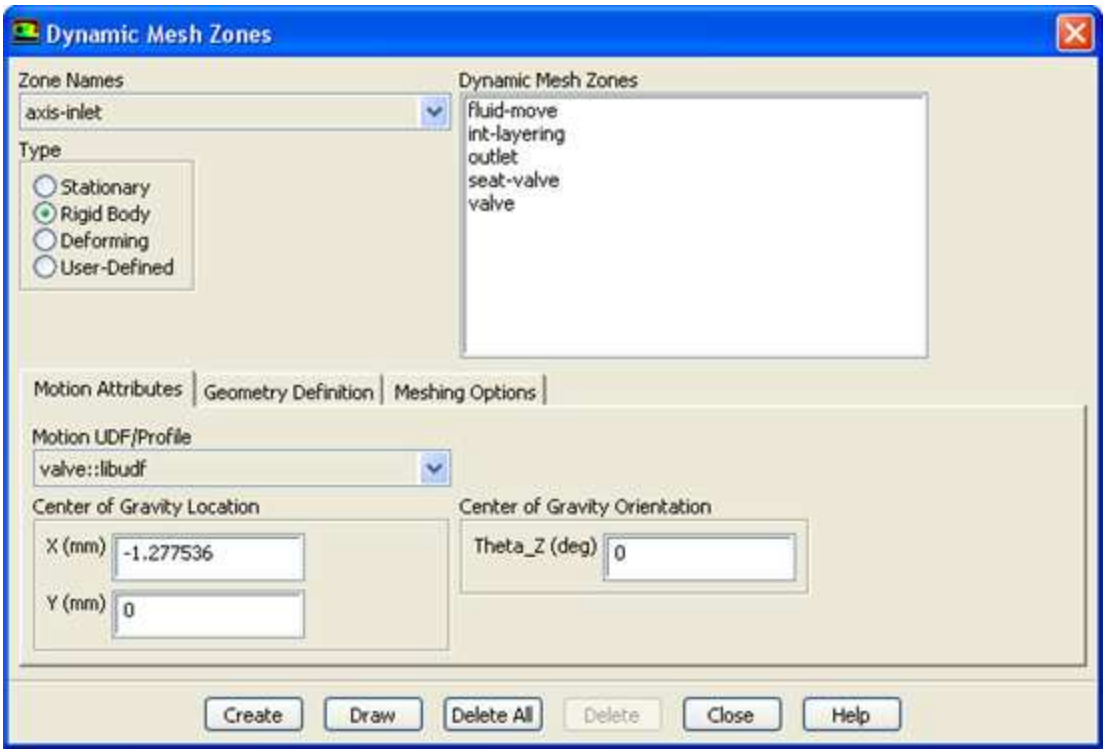
Increment
specifies the size of the step to take during the preview.

Preview
plays back the events at the crank angle specified for each event and reports when each event occurs in the text (console) window.

35.9.8. Dynamic Mesh Zones Dialog Box

The **Dynamic Mesh Zones** dialog box allows you to specify the motion of the dynamic zones in your model. See *Specifying the Motion of Dynamic Zones* (p. 613) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Names

contains the names of the zones in the model.

Type

contains the types of motion that can be specified for a dynamic zone.

Stationary

explicitly declares the zone as stationary, so that the nodes on this zone are excluded when updating the node positions. See *Specifying the Motion of Dynamic Zones* (p. 613) for more information.

Rigid Body

specifies the zone as having a rigid-body motion.

Deforming

specifies the zone as deforming.

User-Defined

enables you to specify a user-defined zone motion.

Dynamic Mesh Zones

lists all of the dynamic zones in the case.

Motion Attributes

contains parameters to specify the motion attributes for a rigid-body-motion zone and a user-defined-motion zone.

Motion/UDF Profile

specifies the motion of the rigid body zone by selecting a profile or user-defined function from the See *Profiles* (p. 366) and *Solid-Body Kinematics* (p. 623) for information on profiles, and see the UDF Manual for information on user-defined functions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Six DOF UDF

specifies the motion of a rigid body zone by selecting a profile or user-defined function from the drop-down list. This option is available only if the **Six DOF** solver is the selected model.

Six DOF Options
contains parameters for the six DOF solver.

On
enables the use of the six DOF solver.

Passive
(if enabled) does not take forces and moments on the zone into consideration.

Center of Gravity Location
contains the current values of the coordinates for the location of the C.G. of the selected zone. This item will appear only if the motion **Type** is **Rigid Body**, if **In-Cylinder** is not enabled in the **Dynamic Mesh Parameters** dialog box.

Center of Gravity Orientation
contains the current values of the coordinates for the orientation defined at the C.G. of the selected zone. This item will appear only if the motion **Type** is **Rigid Body**, if **In-Cylinder** is not enabled in the **Dynamic Mesh Parameters** dialog box.

Center of Gravity Velocity

specifies the velocity of the center of gravity with respect to inertia coordinate system. This option is available only if the **Six DOF** solver is the selected model.

Center of Gravity Angular Velocity

specifies the angular velocity of the center of gravity with respect to inertia coordinate system. This option is available only if the **Six DOF** solver is the selected model.

Lift/Stroke
contains the current value of valve lift or piston stroke which is automatically updated when you click **Create**. This item will appear only if **In-Cylinder** is enabled in the **Dynamic Mesh Parameters** dialog box.

Valve/Position Axis
specifies the direction of the reference axis of the valves or piston for an in-cylinder problem. This item will appear only if **In-Cylinder** is enabled in the **Dynamic Mesh** task page.

Mesh Motion UDF
allows you to select the user-defined function that defines the geometry and motion of the zone. This item will appear only if the motion **Type** is **User-Defined**.

Geometry Definition
contains parameters to define a deforming zone.

Definition
allows you to select a geometry definition to project nodes of the deforming zone on. Available options are **faceted**, **plane**, **cylinder**, and **user-defined**.

Feature Detection
allows you to preserve features on deforming zones between the different face zones and within a face zone.

Include Features
includes features of a specific angle.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature Angle
specifies feature angle in degrees.

Point on Plane
allows you to specify the position of a point on the plane. This item will appear only when you select **plane** for the **Definition**.

Plane Normal
allows you to specify the direction of the plane normal. This item will appear only when you select **plane** for the **Definition**.

Cylinder Radius
allows you to specify the radius of the cylinder. This item will appear only when you select **cylinder** for the **Definition**.

Cylinder Origin
allows you to specify the location of the cylinder origin. This item will appear only when you select **cylinder** for the **Definition**.

Cylinder Axis
allows you to specify the direction of the cylinder axis. This item will appear only when you select **cylinder** for the **Definition**.

Geometry UDF
allows you to select a user-defined function for a geometry **Definition**.

Meshing Options
contains parameters for various meshing options.

Adjacent Zone
contains the name of the adjacent zone that is involved in local remeshing or dynamic layering. This item will appear only if the zone type is **Stationary**, **Rigid Body**, or **User-Defined**, and if the zone is not the boundary of a CutCell dynamic cell zone.

Cell Height
allows you to specify the ideal height of the adjacent cells as either a constant value or a compiled user-defined function. This item will appear only if the motion **Type** is **Stationary**, **Rigid Body**, or **User-Defined**, and if the zone is not the boundary of a CutCell dynamic cell zone.

Methods
contains options to specify the mesh update method(s) and controls on a zone by zone basis. These options and parameters will override those that you specified globally in the **Dynamic Mesh** task page. This item will appear only if the motion **Type** is **Deforming**.

Smoothing
enables the smoothing mesh update method.

Layering
enables the layering mesh update method (valid for a prismatic cell zone only). This item will appear only if the zone you are defining is a cell (fluid) zone.

Remeshing
enables the remeshing mesh update method.

Zone Parameters
contains a set of remeshing criteria, other than those you specified globally in the **Mesh Method Settings** dialog box. Note that this group box is not available if the **CutCell** option in the **Remeshing Options** group box is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Length Scale
allows you to specify the minimum length scale for the zone. This item will appear only if the zone you are defining is a cell (fluid) zone.

Maximum Length Scale
allows you to specify the maximum length for the zone. This item will appear only if the zone you are defining is a cell (fluid) zone.

Maximum Skewness
allows you to specify the desired value for maximum skewness.

Zone Scale Info...
opens the *Zone Scale Info Dialog Box* (p. 1894), in which you can view the statistics of the mesh, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

CutCell Zone Parameters
contains a set of global parameters used for CutCell zone remeshing. This group box will only appear if the zone selected in the **Dynamic Mesh Zones** list is a cell zone, and if the **CutCell** option is enabled in the **Remeshing Options** group box.

Maximum Mesh Size
allows you to specify the maximum mesh size of the Cartesian CutCell mesh.

Growth Rate
allows you to specify the default growth rate used for mesh refinement by size functions. This growth rate is used during remeshing by the mesh-based size functions of all boundaries of the CutCell cell zone that are not defined as dynamic mesh zones.

Minimum Orthogonal Quality
allows you to specify the minimum allowable orthogonal quality for the cell zone. CutCell remeshing will occur if the orthogonal quality for any cell in the cell zone drops below this value.

Remeshing Interval
specifies the interval of time steps at which the cell zone remeshing will take place, even if the mesh quality has not deteriorated below the value entered for **Minimum Orthogonal Quality**. Enter a value of `0` for the **Remeshing Interval** if you want the remeshing to occur only if there is insufficient mesh quality.

Boundary Zones Info...
opens the *CutCell Boundary Zones Info Dialog Box* (p. 1894), which lists all of the boundary zones for the CutCell dynamic cell zone, along with the remeshing parameters.

Zone Scale Info...
opens the *Zone Scale Info Dialog Box* (p. 1894), in which you can view the statistics of the mesh, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

CutCell Boundary Zone Parameters
contains a set of global parameters used for CutCell zone remeshing. This group box will only appear if the zone selected in the **Dynamic Mesh Zones** list is a boundary of a CutCell dynamic cell zone.

Maximum Mesh Size
allows you to specify the maximum mesh size used by the soft size function. If a value of `0` is entered, a mesh-based size function is used to locally refine the Cartesian mesh near the boundary.

Growth Rate
allows you to specify the local growth rate used by the size function.

Zone Scale Info...
opens the *Zone Scale Info Dialog Box* (p. 1894), in which you can view the statistics of the mesh, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smoothing Methods

contains two methods of smoothing. This option is not available if diffusion-based smoothing has been selected in the **Mesh Method Settings** dialog box.

Spring

enables spring-based smoothing method.

Laplace

enables Laplacian smoothing method.

Remeshing Methods

contains two methods of remeshing.

Region

enables region-based remeshing method.

Local

enables local remeshing method.

Remeshing Options

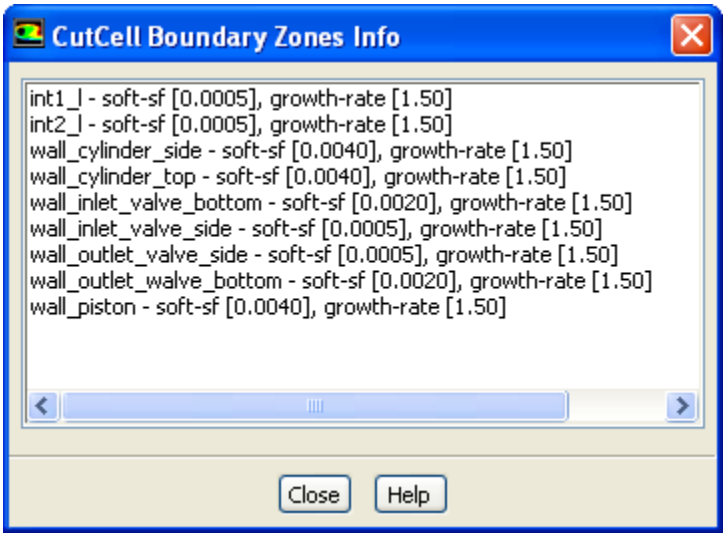
contains one method of remeshing.

CutCell

enables the CutCell zone remeshing method. This option is only available when the **CutCell Zone** is enabled in the **Mesh Method Settings** dialog box, and if the zone selected in the **Dynamic Mesh Zones** list of the **Dynamic Mesh Zones** dialog box is a cell zone.

35.9.9. CutCell Boundary Zones Info Dialog Box

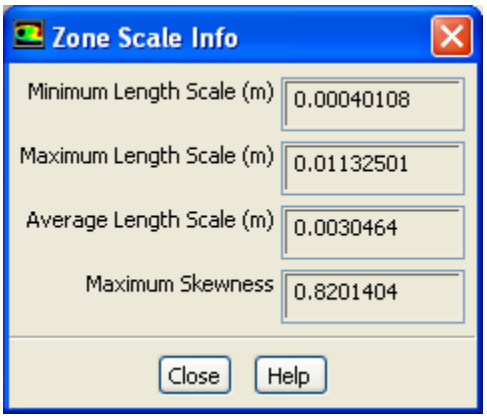
The **CutCell Boundary Zones Info** dialog box displays a list of all of the boundary zones for the CutCell dynamic cell zone. This list also provides the remeshing parameters for the zones, including size function type, the maximum mesh size, and the growth rate used to refine the mesh at all of the CutCell boundaries.



35.9.10. Zone Scale Info Dialog Box

The **Zone Scale Info** dialog box allows you to inspect the values of minimum and maximum cell area or volume, and maximum cell skewness in a zone.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Maximum Length Scale
displays the maximum cell length in the zone.

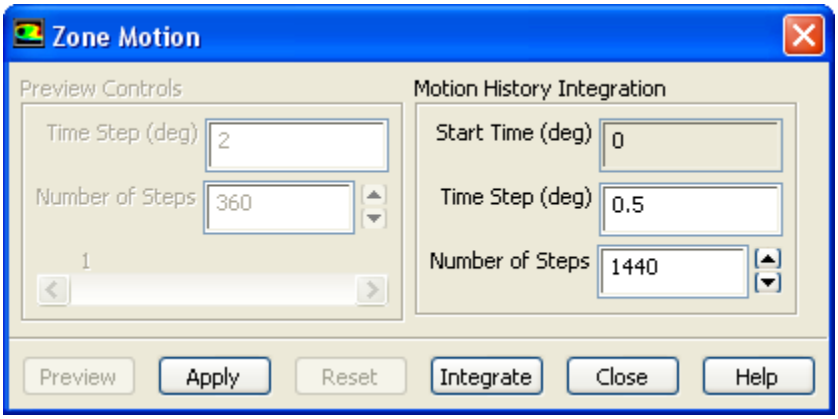
Minimum Length Scale
displays the minimum cell length in the zone.

Average Length Scale
displays the average cell length in the zone.

Maximum Skewness
displays the maximum cell skewness in the zone.

35.9.11. Zone Motion Dialog Box

The **Zone Motion** dialog box will update those zones that have solid body motion specified, in the graphics window. See *Previewing the Dynamic Mesh* (p. 626) for details about the items below.



Controls

Preview Controls
contains controls that you can set to preview the zone motion.

Time Step
specifies time step size for zone motion preview.

Number of Steps
specifies number of time steps for zone motion preview.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Motion History Integration
contains motion history integration parameters.

Start Time
is the starting time for mesh motion (default value set to 0).

Time Step
specifies the time step size in seconds.

Number of Steps
allows you to specify the number of time steps for the motion.

Preview
previews the zone motion.

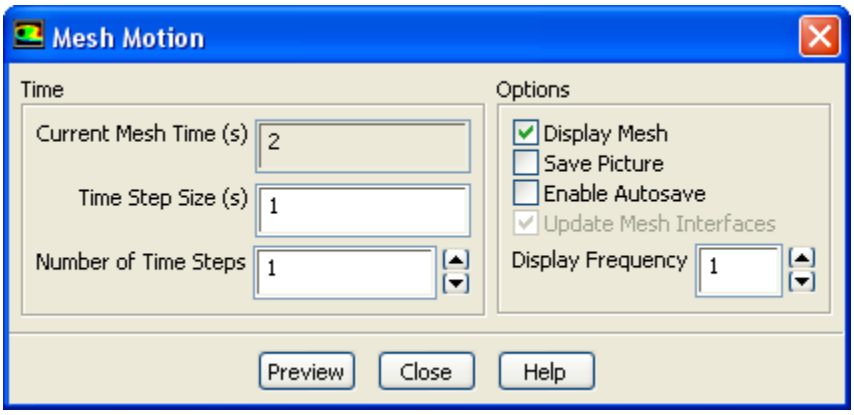
Apply
saves your settings for zone motion.

Reset
sets the default inputs restored in the dialog box.

Integrate
allows ANSYS FLUENT to create a table of surface positions in time.

35.9.12. Mesh Motion Dialog Box

The **Mesh Motion** dialog box allows you to preview the dynamic mesh as it changes with time before you start your simulation. See *Previewing the Dynamic Mesh* (p. 626) for details.



Controls

Time
contains the parameters to specify the time interval at which to update the mesh.

Current Mesh Time
displays the current time after the dynamic mesh has been advanced the specified number of steps.

Time Step Size
specifies the size of each time step.

Number of Time Steps
specifies the number of time steps.

Options
contains options to view the updated mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Mesh
displays the mesh.

Save Picture
opens the *Save Picture Dialog Box* (p. 1988), allowing you to save a picture file of the mesh each time ANSYS FLUENT updates it during the mesh preview.

Enable Autosave
opens the *Autosave Case During Mesh Motion Preview Dialog Box* (p. 1897), allowing you to save the case and data files with the specified name and frequency. See *Automatic Saving of Case and Data Files* (p. 65) for details.

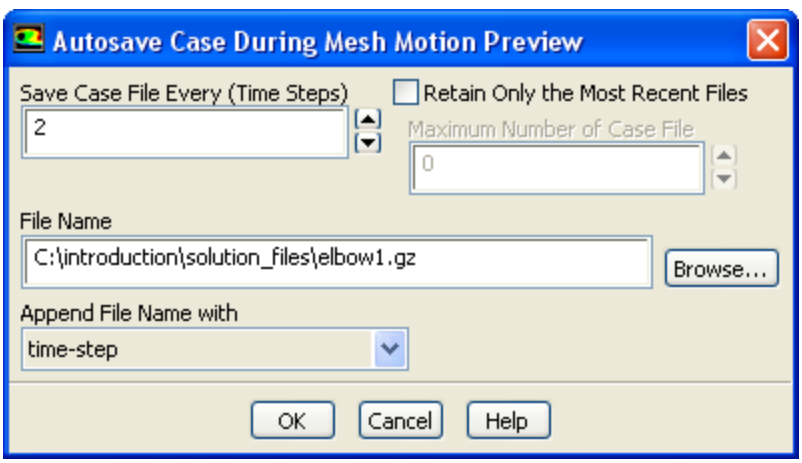
Update Mesh Interfaces
allows you to update the interface at every time step.

Display Frequency
displays the frequency at which ANSYS FLUENT will update the mesh display.

Preview
allows you to preview the motion of the selected zones in the graphics window.

35.9.13. Autosave Case During Mesh Motion Preview Dialog Box

The **Autosave Case During Mesh Motion Preview** dialog box allows you to automatically save your case during the dynamic mesh preview as it changes with time, while running your simulation. See *Automatic Saving of Case and Data Files* (p. 65) for details about the inputs.



Controls

Save Case File Every
contains the parameters to specify the time interval at which to update the mesh.

Retain Only the Most Recent Files
allows you to restrict the number of files saved by ANSYS FLUENT if you have limited disk space. When this option is enabled, you can enter the appropriate value in the **Maximum Number of Data Files** field.

Note

Only the associated case files are retained when using this option.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Data Files

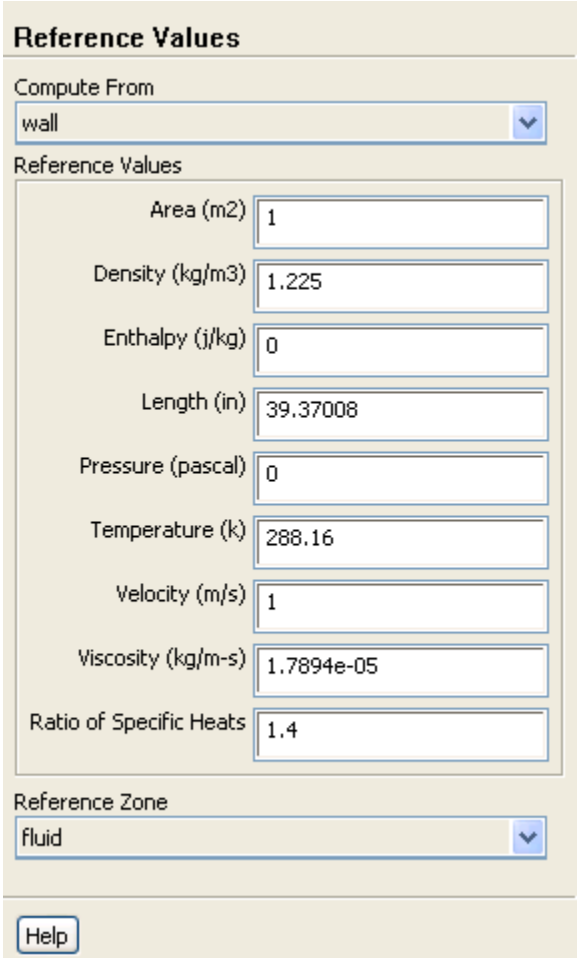
specifies the maximum number of data files that can be saved at any instance. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, ANSYS FLUENT will overwrite the earliest existing file. The default value for this field is zero which saves all the files.

Append File Name with

allows you to select **flow-time** or **time-step** to be appended to the file name. This option is available only for unsteady-state calculations. The default selection is **flow-time**.

35.10. Reference Values Task Page

The **Reference Values** task page allows you to set the reference quantities used for computing normalized flow field variables. It also allows you to specify the reference zone for postprocessing relative velocities in moving-zone problems. See *Reference Values* (p. 1523) for details about the items below.



Controls

Compute from

contains a drop-down list of the boundary zones. You may select a zone to be used for automatically defining the reference values, but depending on the boundary condition used, all of the reference values may not be set. For example, the reference length and area will not be set by computing the reference values from a boundary condition; you will need to set these manually.

Reference Values

contains inputs for the reference values.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Area
sets the reference area, which is used to compute the force and moment coefficients.

Depth
sets the reference depth used for computing cell volumes in 2D.

Density
sets the reference density, which is used to compute the reference dynamic pressure.

Enthalpy
sets the reference enthalpy, which is used to determine the total enthalpy change.

Length
sets the reference length, which is used in the computation of the moment coefficient.

Pressure
sets the reference pressure, which is used to compute the pressure-related forces and moments and the pressure coefficient.

Temperature
sets the reference temperature, which is used to compute entropy for incompressible flows.

Velocity
sets the reference velocity magnitude, which is used to compute the reference dynamic pressure.

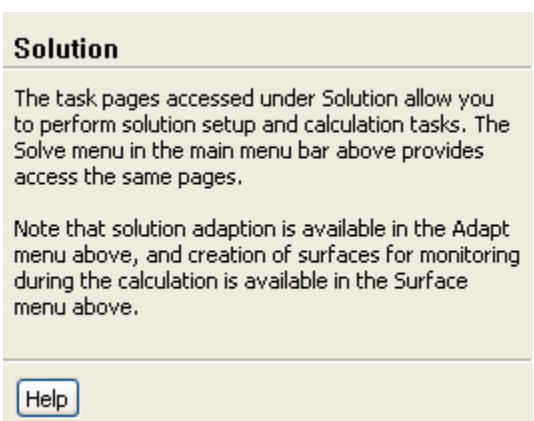
Viscosity
sets the reference kinematic viscosity, which is used in the computation of the boundary Reynolds number.

Ratio of Specific Heats
sets the value of the specific heat ratio, which is used in turbomachinery efficiency calculations.

Reference Zone
contains a drop-down list of all cell zones in the domain. For flows involving multiple moving zones, you will need to select the reference zone for postprocessing relative velocities and related quantities. See *Setting the Reference Zone* (p. 1525) for details.

35.11. Solution Task Page

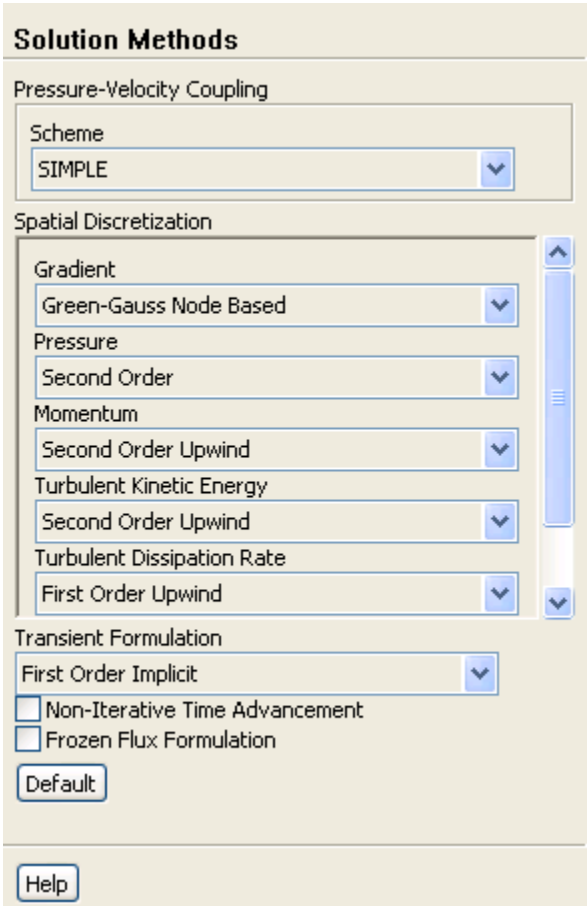
The **Solution** task page introduces you to the main tasks involved in solving your CFD simulation using ANSYS FLUENT. For more information about using ANSYS FLUENT to solve your CFD simulation, see *Using the Solver* (p. 1207).



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.12. Solution Methods Task Page

The **Solution Methods** task page allows you to specify various parameters associated with the solution method to be used in the calculation.



Controls

Formulation

provides a drop-down list of the available types of solver formulations: **Implicit** and **Explicit**.

This item appears only when the density-based solver is used.

Flux Type

provides a drop-down list of the convective flux types: **Roe-FDS**, **AUSM**, and **Low Diffusion Roe-FDS**. Details about each of the flux types can be found in Convective Fluxes in the Theory Guide.

Pressure-Velocity Coupling

contains settings for pressure-velocity coupling schemes.

Scheme

provides a drop-down list of the available pressure-velocity coupling schemes: **SIMPLE**, **SIMPLEC**, **PISO**, and **Coupled**. **Fractional Step** is available in the drop-down list when the non-iterative time advancement (NITA) scheme is enabled in the **Solution Methods** task page. See Pressure-Velocity Coupling in the Theory Guide and *Choosing the Pressure-Velocity Coupling Method* (p. 1215) and *Setting Solution Controls for the Non-Iterative Solver* (p. 1219) for details about these methods.

This item appears only when the pressure-based solver is used.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skewness Correction
enables the SIMPLEC and PISO skewness correction for highly skewed meshes if the value (number of iterations) is greater than 0. The default value is 0 for SIMPLEC and 1 for PISO.

Neighbor Correction
enables the PISO neighbor correction, which is recommended for transient calculations, if the value (number of iterations) is greater than 0. The default value is 1.

Skewness-Neighbor Coupling
allows for a more economical but a less robust variation of the PISO algorithm.

Spatial Discretization
contains settings that control the spatial discretization of the convection terms in the solution equations. See *Choosing the Spatial Discretization Scheme* (p. 1210) for details.

Gradient
contains a drop-down list of the options for setting the method of computing the gradient in Equation 19–28 in the Theory Guide: **Green-Gauss Cell Based**; **Green-Gauss Node-Based**, and **Least Squares Cell Based**. See Evaluation of Gradients and Derivatives in the Theory Guide for details.

Pressure
(for the pressure-based solver only) contains a drop-down list of the discretization schemes available for the pressure equation: **Standard**, **PRESTO!**, **Linear**, **Second Order**, and **Body Force Weighted**.

This item appears only when the pressure-based solver is used.

Flow
(for the density-based solvers only) contains a drop-down list of the discretization schemes available for the pressure, momentum, and (if relevant) energy equations: **First Order Upwind**, **Second Order Upwind**, and **Third-Order MUSCL**.

This item appears only when one of the density-based solvers is used.

Momentum, Energy, etc.
are the names of the other convection-diffusion equations being solved. In the drop-down list next to each equation, you can select the **First Order Upwind**, **Second Order Upwind**, **Power Law**, **QUICK**, or **Third-Order MUSCL** discretization scheme for that equation.

If the LES turbulence model is enabled, then you have a choice of selecting **Bounded Central Differencing** or **Central Differencing**to solve the convection-diffusion equations.

If one of the density-based solvers is used, **Momentum** and **Energy** will not appear. For the density-based solvers, the discretization scheme for these equations is selected in the **Flow** drop-down list (described above).

Volume Fraction
is available when the VOF multiphase model is enabled. The discretization schemes that are used when solving volume fraction equations for the VOF explicit scheme are **Geo-Reconstruct**, **Compressive**, **CICSAM**, **Modified HRIC**, and **QUICK**. The discretization schemes that are used when solving volume fraction equations for the VOF implicit scheme are **First Order Upwind**, **Second Order Upwind**, **Compressive**, **Modified HRIC**, **BGM** (steady state only), and **QUICK**. See Interpolation Near the Interface in the Theory Guide for detailed information about these VOF-specific interpolation schemes.

Transient Formulation
contains options for setting different time-dependent solution formulations. This option appears only when **Transient** is enabled under **Time** in the **General** task page. Available options include: **Explicit**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(available only for the **Density Based Explicit** solver); **First Order Implicit**; **Second Order Implicit**; and **Bounded Second Order Implicit**. See *Performing Time-Dependent Calculations* (p. 1255) for details.

Non-iterative Time Advancement

enables non-iterative time-advancement (NITA) scheme. See Time-Advancement Algorithm in the Theory Guide for details.

Frozen Flux Formulation

enables an option to discretize the convective part of Equation 19–61 in the Theory Guide using the mass flux at the cell faces from the previous time level n. This option is available only for a **Transient** solution. See The Frozen Flux Formulation in the Theory Guide for details.

Pseudo Transient

enables an option to apply the pseudo transient under-relaxation method, which is a form of implicit under-relaxation (see Pseudo Transient Under-Relaxation). This option is available for the pressure-based solver when **Coupled** is selected as the **Pressure-Velocity Coupling** scheme and for the density-based implicit solver. Note that this method can only be used when running a steady-state simulation.

Set All Species Discretizations Together

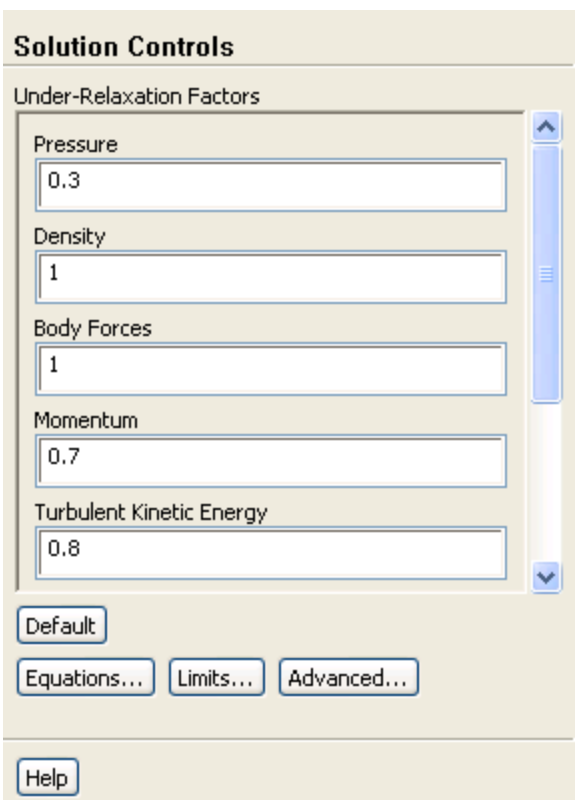
enables an option to use the same discretization scheme for all the species rather than setting each of the species individually. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the scheme of your choice. Note that this option is available when species transport is enabled.

Default

sets the fields to their default values, as assigned by ANSYS FLUENT.

35.13. Solution Controls Task Page

The **Solution Controls** task page allows you to set common solution parameters.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Courant Number

sets the fine-grid Courant number (time step factor) when the density-based solver is used. See *Changing the Courant Number* (p. 1222) for guidelines on setting the Courant number.

When the pressure-based solver is used for time-independent flows, and the **Coupled** pressure-velocity scheme is used, the **Courant Number** is used to stabilize the convergence behavior. See Under-Relaxation of Equations in the Theory Guide for a correlation of the under-relaxation factor and courant number.

Explicit Relaxation Factors

for the **Coupled** scheme defines the explicit relaxation of variables between sub-iterations for momentum and pressure. See Under-Relaxation of Variables in the Theory Guide for information.

Multigrid Levels

specifies the maximum number of coarse levels to be created by the FAS multigrid solver. This item is the same as the **Max Coarse Levels** under **FAS Multigrid Controls** in the **Multigrid** tab of the *Advanced Solution Controls Dialog Box* (p. 1906), and it appears only when the density-based explicit solver is used.

Residual Smoothing

contains parameters that govern the use of implicit residual smoothing. (See Implicit Residual Smoothing in the Theory Guide and *Using Residual Smoothing to Increase the Courant Number* (p. 1226) for details.) This section of the task page will appear only when the density-based explicit solver is used.

Iterations

sets the number of iterations of the Jacobi smoother to use. If **Iterations** is 0, then no implicit residual smoothing is performed.

Smoothing Factor

sets the implicit residual smoothing factor. This item will not appear unless **Iterations** is set to a non-zero value.

Under-Relaxation Factors

contains the under-relaxation factors for all equations that are being solved with the pressure-based solver. (See *Setting Under-Relaxation Factors* (p. 1217) for details.) In the field next to each equation, you can set the under-relaxation factor for that equation.

When one of the density-based solvers is used, **Under-Relaxation Factors** will appear only for the following variables, when they are included in your model: solid (for conjugate heat transfer models), turbulence variables, and viscosity. The density-based solvers use a segregated method to solve these equations; all the other equations are solved in a coupled manner, so there are no under-relaxation factors for them.

When **Non-iterative Time Advancement** is selected for the pressures-based solver, the **Non-Iterative Solver Relaxation Factors** define the explicit relaxation (Under-Relaxation of Variables in the Theory Guide) of variables between sub-iterations and are used to prevent the solution from diverging.

When **Pseudo Transient** is selected (available with the pressures-based coupled solver), the **Pseudo Transient Explicit Relaxation Factors** can be specified (see *Setting Pseudo Transient Explicit Relaxation Factors* (p. 1250)).

Default

sets the fields to their default values, as assigned by ANSYS FLUENT.

Equations...

displays the *Equations Dialog Box* (p. 1904).

Limits...

displays the *Solution Limits Dialog Box* (p. 1904).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced...
displays the *Advanced Solution Controls Dialog Box* (p. 1906).

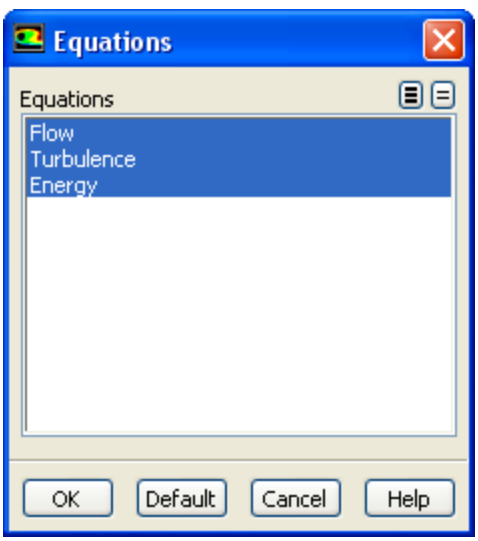
Set All Species URFs Together
enables an option to use the same under-relaxation factors for all the species rather than setting each of the species individually. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the specified under-relaxation factor. Note that this option is available when species transport is enabled.

For additional information, please see the following sections:
35.13.1. Equations Dialog Box
35.13.2. Solution Limits Dialog Box
35.13.3. Advanced Solution Controls Dialog Box

35.13.1. Equations Dialog Box

The **Equations** dialog box contains a list of the equations being solved for the current model. To temporarily disable solution of an equation, deselect it in this list and click **OK**. To re-enable the calculation for an equation, select it and click **OK**. See *Step-by-Step Solution Processes* (p. 1318) for details about using this feature in a step-by-step solution process.

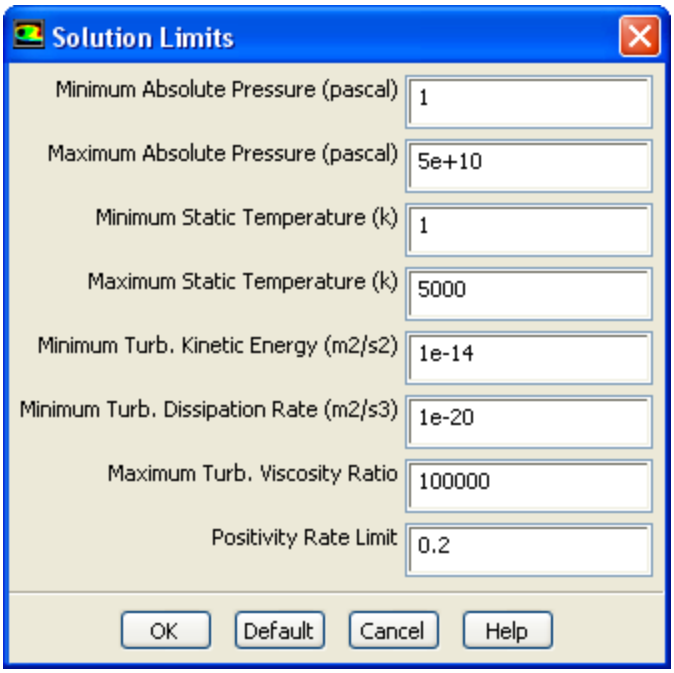
Note that, when one of the density-based solvers is used, **Energy** will not appear as a separate item in the **Equations** list. For the density-based solvers the energy equation is included in the **Flow** category (which also includes the pressure and momentum equations).



35.13.2. Solution Limits Dialog Box

The **Solution Limits** dialog box allows you to improve the stability of the solution. See *Setting Solution Limits* (p. 1235) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Minimum Absolute Pressure, Maximum Absolute Pressure
set the limiting minimum and maximum allowable values for absolute pressure.

Minimum Static Temperature, Maximum Static Temperature
set the limiting minimum and maximum allowable values for temperature.

Minimum Turb. Kinetic Energy
sets the limiting minimum value of turbulent kinetic energy (k) in the flow field. This parameter appears when one of the k- ε or k- ω models or the RSM is used.

Minimum Turb. Dissipation Rate
sets the limiting minimum value of turbulent dissipation rate (ε) in the flow field. This parameter appears when one of the k- ε models or the RSM is used.

Minimum Spec. Dissipation Rate
sets the limiting minimum value of specific dissipation rate (ω) in the flow field. This parameter appears when one of the k- ω models is used.

Maximum Turb. Viscosity Ratio
sets the limiting maximum allowable value of the ratio of turbulent to laminar viscosity (μ_t/μ). The turbulent viscosity is reduced to the necessary value so as not to exceed the maximum allowable viscosity ratio.

This parameter appears for all turbulent flows.

Positivity Rate Limit
sets the limiting value for the rate of reduction of temperature. See *Adjusting the Positivity Rate Limit* (p. 1236) for details.

This item appears only when one of the density-based solvers is used.

Default
sets the fields to their default values, as assigned by ANSYS FLUENT. After execution, the **Default** button becomes the **Reset** button.

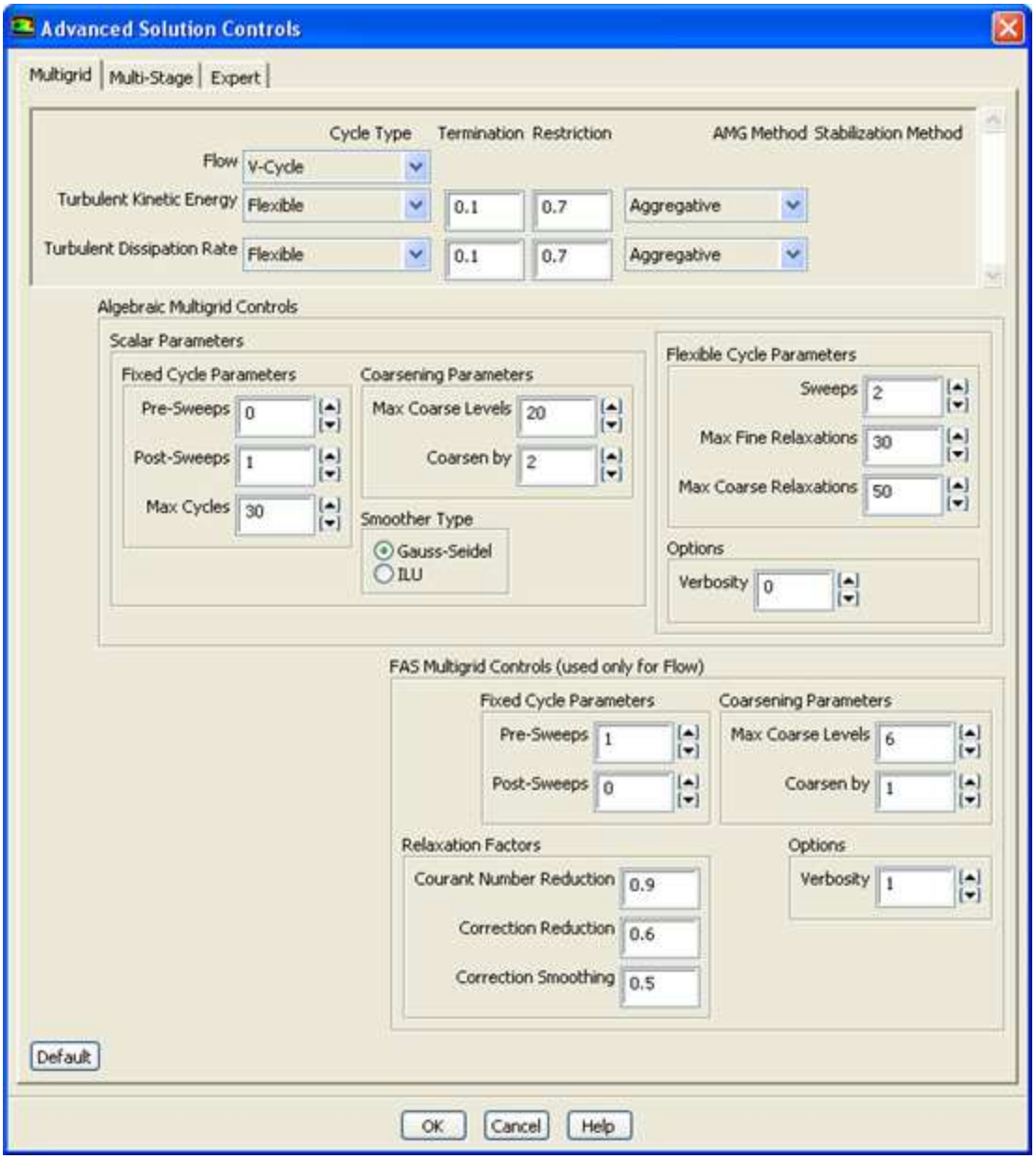
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset

resets the fields to their most recently saved values (e.g., the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

35.13.3. Advanced Solution Controls Dialog Box

The **Advanced Solution Controls** dialog box allows you to set parameters related to the multigrid, multi-stage, and non-iterative solvers.



Controls

Multigrid

tab contains parameters related to the multigrid solver. See *Modifying Algebraic Multigrid Parameters* (p. 1319) and *Setting FAS Multigrid Parameters* (p. 1232) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cycle Type

contains a drop-down list for each equation that is being solved. From this list you can select the multigrid cycle type (**Flexible**, **V-Cycle**, **W-Cycle**, or **F-Cycle**). See *Specifying the Multigrid Cycle Type* (p. 1227) for details.

Note that, for the density-based solvers, the **Pressure**, **Momentum**, and **Energy** equations will not appear individually. They will instead be grouped together and called **Flow**.

Furthermore, for the density-based explicit solver, the **Cycle Type** choices for the **Flow** equations will be limited to **V-Cycle** and **W-Cycle**, while the choices for **Flow** for the density-based implicit solver will be limited to **V-Cycle** and **F-Cycle**.

Termination

specifies the termination criterion for each equation that is being solved using algebraic multigrid. See *Setting the Termination and Residual Reduction Parameters* (p. 1227) for details.

Restriction

specifies the residual reduction criterion for each equation that is being solved using the **Flexible** algebraic multigrid cycle. See *Setting the Termination and Residual Reduction Parameters* (p. 1227) for details. (This item will not appear for an equation that is using a **V-Cycle**, **W-Cycle**, or **F-Cycle**.)

AMG Method

contains the drop-down list to choose between two AMG solvers: aggregative or selective. See *Setting the AMG Method and the Stabilization Parameters* (p. 1228) for details.

Aggregative

enables aggregative AMG solver.

Selective

enables selective AMG solver. The selective AMG solver is available only for scalar equations, and is not available in parallel ANSYS FLUENT.

Stabilization Method

contains the drop-down list to choose the stabilization method.

BCGSTAB

enables bi-conjugate gradient stabilized method.

RPM

enables the recursive projection method. RPM stabilization is mainly used in conjunction with the coupled pressure-based solver.

Algebraic Multigrid Controls

contains parameters related to the algebraic multigrid solver. See Algebraic Multigrid (AMG) in the Theory Guide and *Modifying Algebraic Multigrid Parameters* (p. 1319) for details.

Scalar and Coupled Parameters

contain parameters that you can set, If you are using the density-based explicit solver or the pressure-based solver with any of the segregated schemes, described in Pressure-Velocity Coupling in the Theory Guide and *Choosing the Pressure-Velocity Coupling Method* (p. 1215), you will only set **Scalar Parameters**. If you are using the density-based implicit or the pressure-based coupled scheme, described in Coupled Algorithm, then you can set the **Coupled Parameters**.

Fixed Cycle Parameters

contains parameters that control the V, W, and F cycles. You can set the number of **Pre-Sweeps** and **Post-Sweeps**, and the **Max Cycles**. Normally one post-sweep is performed and no pre-sweeps are done. See *Fixed Cycle Parameters* (p. 1229) for details about using the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pre-Sweeps
sets the number of sweeps to perform before moving to a coarser level.

Post-Sweeps
sets the number of sweeps to perform after coarser level corrections have been applied.

Max Cycles
sets the maximum number of V, W, or F cycles to be performed. The multigrid solver will continue to solve the set of equations until either the maximum number of cycles has been performed, or the **Termination** criteria are satisfied.

Coarsening Parameters
contains parameters that control the grouping of equations in the algebraic multigrid algorithm. See *Coarsening Parameters* (p. 1229) for details

Max Coarse Levels
is the maximum number of coarse levels that will be built by the multigrid solver. Sets of coarser simultaneous equations are built until the maximum number of levels has been created, or the coarsest level has only 3 equations. Each level has about half as many unknowns as the previous level, so coarsening until there are only a few nodes left will require about as much total coarse-level coefficient storage as was required on the fine mesh. Reducing the maximum coarse levels will reduce the memory requirements, but may require more iterations to achieve a converged solution. Setting **Max Coarse Levels** to 0 turns off the multigrid solver.

Coarsen by
controls the number of equations on each successively coarser grid level. By default, this parameter is set to 2, indicating that the number of equations on each level will be 1/2 of the number on the previous level. In general, the number of equations on each coarser grid level will be equal to 1/ n of the number on the previous level, where n is the value set for the **Coarsen by** parameter.

Smoother Type
consist of two types:

Gauss-Seidel
is the simplest smoother type and is recommended when using the pressure-based segregated solver.

ILU
is more CPU intensive, but has better smoothing properties for block-coupled systems such as the pressure-based coupled solver and the density-based implicit solver.

Flexible Cycle Parameters
contains parameters that control the flexible multigrid cycle.

Sweeps
specifies the number of times to apply the smoothing method each time a relaxation is performed.

Max Fine Relaxations
sets the maximum number of relaxations to be performed on the Level 1 grid (fine grid level). This parameter eliminates the possibility that the Gauss-Seidel solver will get "stuck" on the fine grid level, unable to reduce the residuals by the fraction (α) required by the **Termination** criteria.

Max Coarse Relaxations
sets the maximum number of relaxations to be performed on each grid level above Level 1 (i.e., the coarse grid levels). This parameter eliminates the possibility that the Gauss-Seidel

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solver will get "stuck" on a coarse grid level, unable to reduce the residuals on that level by the fraction (α) required by the **Termination** criteria. If the iterative solution on a given grid level is unable to meet the accuracy constraint of the **Termination** criteria, the correction equation will be deemed "converged" when this maximum number of relaxations on that grid level has been performed.

Options

contains additional multigrid parameters.

Verbosity

controls the amount of information that is printed out by the multigrid solver for monitoring purposes. See *Setting the Verbosity* (p. 1231) for details

FAS Multigrid Controls

contains parameters related to the FAS multigrid solver. See Full-Approximation Storage (FAS) Multigrid in the Theory Guide and *Setting FAS Multigrid Parameters* (p. 1232) for details. As noted in the title of this dialog box section, the FAS multigrid solver is used only for the **Flow** equations (pressure, momentum, and energy).

This section of the dialog box appears only when the density-based explicit solver is used.

Fixed Cycle Parameters

contains parameters that control the V, W, and F cycles of the FAS multigrid solver.

Pre-Sweeps

sets the number of iterations of the multi-stage solver to be performed on a given grid level before proceeding to a coarser grid level (the value of β_1 described in Multigrid Cycles in the Theory Guide). Typically, this is set to 1.

Post-Sweeps

sets the number of multigrid cycles to be performed on a given grid level before proceeding back up to the finer grid level (the value of β_2 described in Multigrid Cycles in the Theory Guide). A value of 1 results in V-cycle multigrid, and a value of 2 results in W-cycle multigrid.

Coarsening Parameters

contains parameters that control the grouping of cells in the FAS multigrid algorithm.

Max Coarse Levels

sets the maximum number of grid levels to be used in the multigrid process. A value of 0 disables multigrid (no coarse grid levels). If the coarse grids do not already exist, they are created automatically when you start iterating; you cannot create them by clicking the **OK** button. See *Turning On FAS Multigrid* (p. 1225) for details.

Coarsen by

controls the number of cells in each successively coarser grid level. By default, this parameter is set to 2, indicating that the number of cells on each level will be 1/2 of the number on the previous level. In general, the number of cells on each coarser grid level will be equal to 1/ n of the number on the previous level, where n is the value set for the **Coarsen by** parameter.

Relaxation Factors

are provided to moderate and stabilize the multigrid corrections.

Courant Number Reduction

sets the factor by which to reduce the Courant number for coarse grid levels (i.e., every level except the finest). Some reduction of time step (such as the default 0.9) is typically required because the stability limit cannot be determined as precisely on the irregularly shaped coarser grid cells.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Correction Reduction

sets the factor by which to reduce the magnitude of the multigrid corrections transferred from one level to the next finer level. A typical value with $\beta_1 = 1$ is 0.6. If two **Pre-Sweeps** and two **Post-Sweeps** are performed, this value can often be increased to 1.0 (i.e., full correction transfer).

Species Correction Reduction

sets the factor by which to reduce the magnitude of the species corrections to stabilize the multigrid calculation. This item appears only when species transport is being modeled.

Correction Smoothing

sets the correction smoothing factor used to interpolate corrections from a coarser grid to a finer grid. For multigrid on structured meshes, corrections can be interpolated up to a finer mesh "smoothly" by using, for example, tri-linear interpolation. For unstructured meshes there is no analogous simple, algebraic procedure that can be used to interpolate without introducing substantial high frequency "noise". Instead, the corrections are first interpolated, and then subjected to a smoothing pass. The default **Correction Smoothing** value of 0.5 should be acceptable for all cases; you should not need to change it.

Options

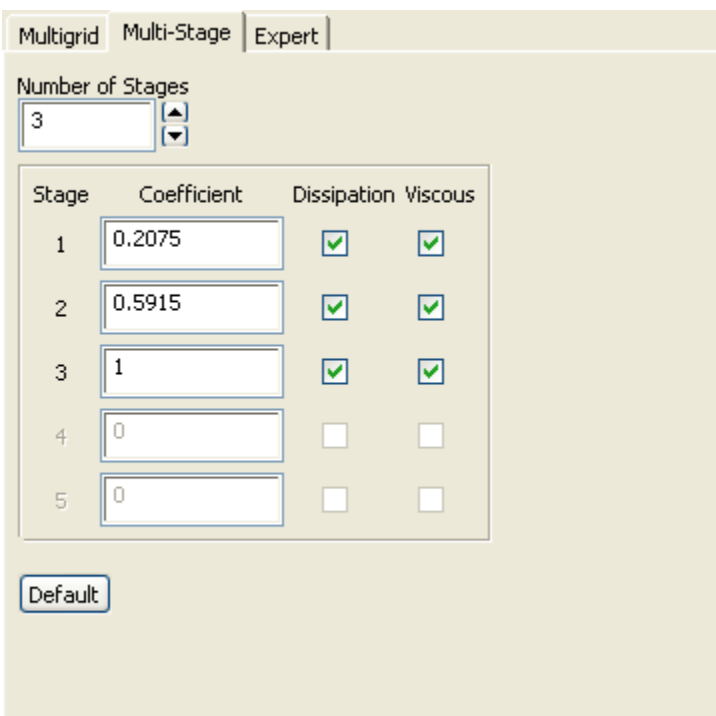
contains additional multigrid parameters.

Verbosity

controls the amount of information that is printed out by the multigrid solver for monitoring purposes.

Mulit-Stage

tab allows you to set parameters that govern the operation of the multi-stage solver. It is available only when the density-based explicit solver is used. See *Changing the Multi-Stage Scheme* (p. 1237) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Stages

is the number of stages used in the multi-stage scheme. The default scheme is a 3-stage scheme with coefficients of 0.2075, 0.5915, and 1.0 for the first through third stages, respectively. Although the dialog box limits the maximum number of stages to five, you can define a scheme with an arbitrary number of stages with the `solve/set/multi-stage` text command.

Stage

labels the stage to which the parameters in the other columns apply.

Coefficient

sets the multi-stage coefficient for each stage. Coefficients should be greater than zero and less than one. The final stage should always have a coefficient of 1.

Dissipation

sets the stages for which artificial dissipation is evaluated. If a **Dissipation** box is selected for a particular stage, artificial dissipation will be updated on that stage. If not selected, artificial dissipation will remain "frozen" at the value of the previous stage.

Viscous

sets the stages for which viscous stresses are evaluated. If a **Viscous** box is selected for a particular stage, viscous stresses will be updated on that stage. If not selected, viscous stresses will remain "frozen" at the value of the previous stage. Viscous stresses should always be computed on the first stage, and successive evaluations will increase the "robustness" of the solution process, but will also increase the expense (i.e., increase the CPU time per iteration). For steady problems, the final solution is independent of the stages on which viscous stresses are updated.

Default

sets the fields to their default values, as assigned by ANSYS FLUENT. After execution, the **Default** button becomes the **Reset** button.

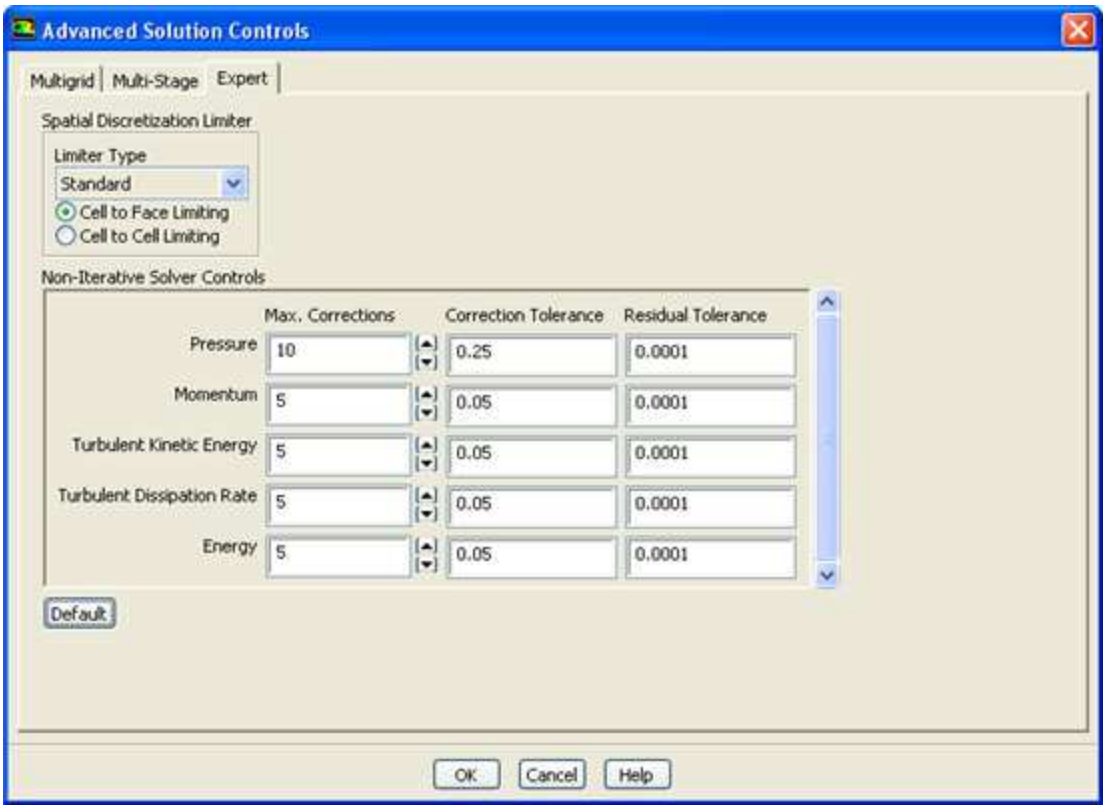
Reset

resets the fields to their most recently saved values (i.e., the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

Expert

tab contains specialized parameters for limiting spatial discretization, as well as controls for the non-iterative solver.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Spatial Discretization Limiter

contains a **Limiter Type** drop-down list of the options available for limiting the spatial discretization: **Standard**, **Multidimensional**, or **Differentiable**. See *Selecting Gradient Limiters* (p. 1239) for more information about limiters.

Cell to Face Limiting

is where the limited value of the reconstruction gradient is determined at the cell face centers. This is the default method.

Cell to Cell Limiting

is where the limited value of the reconstruction gradient is determined along a scaled line between two adjacent cell centroids. On an orthogonal mesh (or when the cell-to-cell direction is parallel to face area direction), this method becomes equivalent to the default cell to face method. For smooth field variation, cell to cell limiting may provide less numerical dissipation on meshes with skewed cells.

Pseudo Transient Specific Time Step Controls

allows you to select and modify the pseudo time steps for specific equations. See *Setting Solution Controls for the Pseudo Transient Method* (p. 1251) for details.

Activate

selects the equation for which you want to specify a different time step from the global pseudo time step.

Time Step

overrides the **Pseudo Time Step** that is specified in the *Run Calculation Task Page* (p. 1955).

Non-Iterative Solver Controls

contain parameters that control the sub-iterations for the individual equations. See Time-Advancement Algorithm in the Theory Guide for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max. Corrections
provide control over the maximum number of sub-iterations for each individual equation.

Correction Tolerance
defines the overall accuracy.

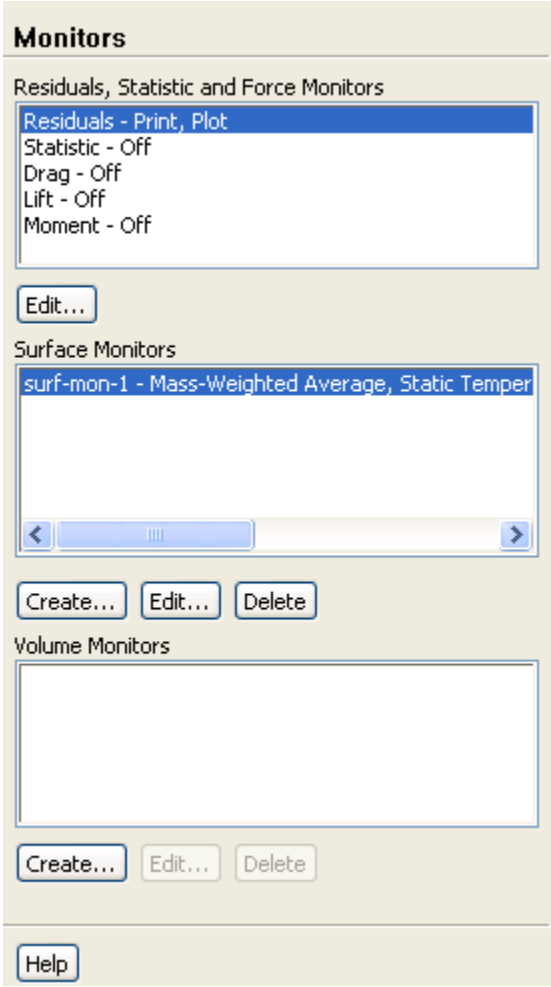
Residual Tolerance
controls the solution of the linear equations.

Default
sets the fields to their default values, as assigned by ANSYS FLUENT. After execution, the **Default** button becomes the **Reset** button.

Reset
resets the fields to their most recently saved values (i.e., the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

35.14. Monitors Task Page

The **Monitors** task page allows you to set up tools for monitoring the convergence of your solution dynamically by checking residuals, statistics, force values, surface integrals, and volume integrals. See *Monitoring Solution Convergence* (p. 1268) for details.



Controls

Residuals, Statistic and Force Monitors

contains a listing of the monitors for your solution residuals, statistics, and/or force values

You can double-click an item in the list to open the corresponding dialog box, or you can select the item in the list and click the **Edit...** button.

Residuals

selecting this item and clicking the **Edit...** button opens the *Residual Monitors Dialog Box* (p. 1915).

Statistic

selecting this item and clicking the **Edit...** button opens the *Statistic Monitors Dialog Box* (p. 1917).

Drag

selecting this item and clicking the **Edit...** button opens the *Drag Monitor Dialog Box* (p. 1918).

Lift

selecting this item and clicking the **Edit...** button opens the *Lift Monitor Dialog Box* (p. 1920).

Moment

selecting this item and clicking the **Edit...** button opens the *Moment Monitor Dialog Box* (p. 1921).

Edit...

displays the dialog box corresponding to the selected item in the **Residuals, Statistic and Force Monitors** list.

Surface Monitors

contains a list of available surface monitors.

Create...

opens the *Surface Monitor Dialog Box* (p. 1923) where you can create a new surface monitor.

Edit...

opens the *Surface Monitor Dialog Box* (p. 1923) where you can edit an existing surface monitor.

Delete

removes the selected surface monitor(s) from the **Surface Monitors** list.

Volume Monitors

contains a list of available volume monitors.

Create...

opens the *Volume Monitor Dialog Box* (p. 1925) where you can create a new volume monitor.

Edit...

opens the *Volume Monitor Dialog Box* (p. 1925) where you can edit an existing volume monitor.

Delete

removes the selected volume monitor(s) from the **Volume Monitors** list.

For additional information, please see the following sections:

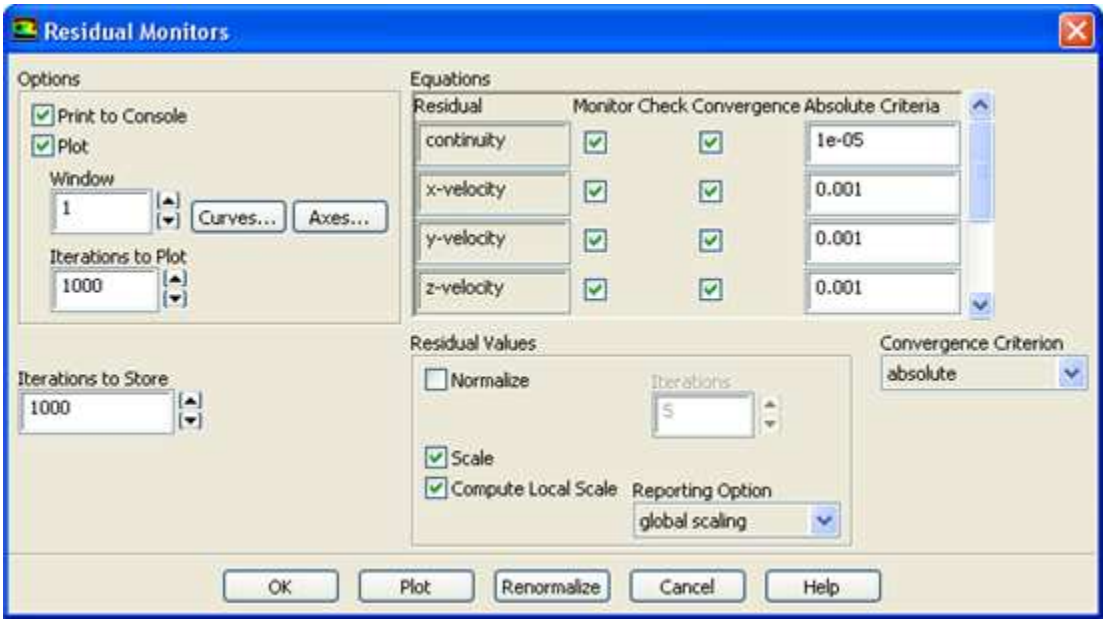
35.14.1. Residual Monitors Dialog Box
35.14.2. Statistic Monitors Dialog Box
35.14.3. Drag Monitor Dialog Box
35.14.4. Lift Monitor Dialog Box
35.14.5. Moment Monitor Dialog Box
35.14.6. Surface Monitor Dialog Box
35.14.7. Volume Monitor Dialog Box
35.14.8. Point Surface Dialog Box
35.14.9. Line/Rake Surface Dialog Box
35.14.10. Plane Surface Dialog Box
35.14.11. Quadric Surface Dialog Box

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.14.12. Iso-Surface Dialog Box
35.14.13. Iso-Clip Dialog Box
35.14.14. Surfaces Dialog Box

35.14.1. Residual Monitors Dialog Box

You can use the **Residual Monitors** dialog box to control the residual information that ANSYS FLUENT reports. See *Monitoring Residuals* (p. 1269) for details about the items below.



Controls

Options

selects any combination of the following methods for reporting residuals. See *Printing and Plotting Residuals* (p. 1272) for details.

Print to Console

specifies whether or not to print residuals in the text window after each iteration.

Plot

specifies whether or not to plot residuals in the graphics window (with the window ID set in **Window**) after each iteration. See *Plot Parameters* (p. 1276) for details.

Window

sets the window ID in which the plot will be drawn. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the residual plot, and then returned to its previous value. Thus, the residual plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

Iterations to Plot

is the number of history points to display on the residual plot.

Axes...

opens the *Axes Dialog Box* (p. 2022), which allows you to modify the attributes of the axes.

Curves...

opens the *Curves Dialog Box* (p. 2024), which allows you to modify the attributes of the residual curves.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iteration to Store

sets the number of residual history points to be stored in the data file. Due to the compaction algorithm used, saving 1000 points does not result in just the last 1000 iterations being saved; the history reaches back quite a bit further than that, but does not save a point at every iteration. Further back in the iteration history, the spacing between saved iterations grows larger. See *Storing Residual History Points* (p. 1273) for details.

Residual Values

controls the normalization and scaling of residuals. See *Controlling Normalization and Scaling* (p. 1273) for details.

Normalize

specifies whether or not to normalize the printed or plotted residual for each variable by the value indicated as the **Normalization Factor** for that variable. The default **Normalization Factor** is the maximum residual value after the first 5 iterations.

This option is off by default.

Iterations

sets the number of iterations for which ANSYS FLUENT will search for the largest residual to normalize by. (If the **Normalize** option is turned off, this item will not be editable.)

Scale

specifies whether or not to print or plot scaled residuals for each variable. This option is on by default.

Compute Local Scale

computes and stores both the locally and globally scaled residuals from subsequent iterations, for the purpose of reporting. You will select the type of residual scaling form the **Reporting Option** drop-down list. See *Definition of Residuals for the Pressure-Based Solver* (p. 1269) and *Definition of Residuals for the Density-Based Solver* (p. 1270) for more information.

Reporting Option

gives you the choice of plotting or printing to the console the **global scaling** or **local scaling** of residuals.

Convergence Criterion

consists of four options that are available for checking an equation for convergence.

absolute

is the default and is available for steady-state cases. The residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a time step.

relative

is where the residual of an equation at an iteration of a time step is compared with the residual at the start of the time step. If the ratio of the two residuals is less than a user-specified value, that equation is deemed to have converged for a time step.

relative or absolute

is where either the **absolute** convergence criterion or the **relative** convergence criterion is met. At that point, the equation is considered converged.

none

is used to disable convergence checking.

Residual

indicates the name of each variable for which residual information is available.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitor

indicates whether or not the residuals for each variable are to be monitored. You can toggle monitoring on and off for each variable by turning the corresponding check box in the **Monitor** list on or off.

Normalization Factor

shows the normalization factor for each variable. The default is the maximum residual value after the first 5 iterations. To set this value manually, enter a new value in the corresponding **Normalization Factor** field. This list will not appear if the **Normalize** option is turned off.

Check Convergence

indicates whether or not the convergence of each variable is to be monitored. If convergence is being monitored, the solution will stop automatically when each variable meets its specified convergence criterion. You can check convergence only for variables for which you are monitoring residuals. You can toggle convergence checking on and off for each variable by turning the corresponding check box in the **Check Convergence** list on or off.

Absolute Criteria

, **Relative Criteria** shows the residual value for which the solution of each variable will be considered converged. To set this value manually, enter a new value in the corresponding **Absolute Criteria**/ **Relative Criteria** field.

Plot

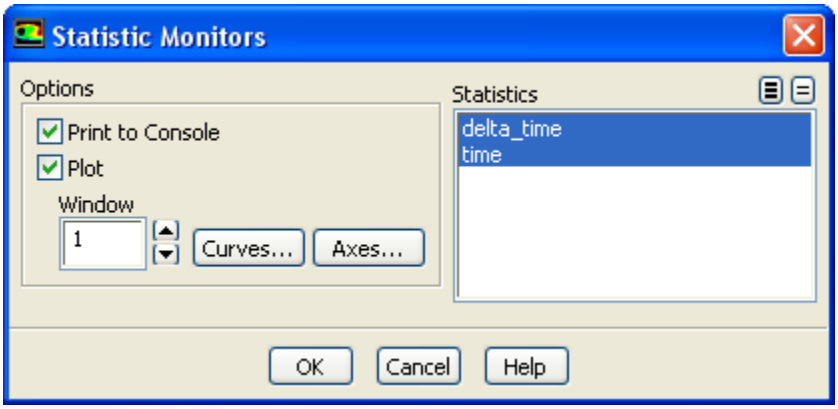
displays the current residual history plot.

Renormalize

sets the normalization factors to the maximum values in the residual histories. **Renormalize** should be used to renormalize the residual plot in cases where the maximum residuals occur sometime after the first five iterations.

35.14.2. Statistic Monitors Dialog Box

You can use the **Statistic Monitors** dialog box to control the statistics information that ANSYS FLUENT reports. See *Monitoring Statistics* (p. 1277) for details about using this dialog box.



Controls

Options

selects the following methods for reporting statistics:

Print to Console

specifies whether or not to print the selected statistics in the console window after each iteration.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot

specifies whether or not to plot each of the selected statistics in a separate graphics window after each iteration. The windows that the plots appear in are determined by the **First Window** option below.

Statistics

contains a list of statistics from which you can select those that are to be plotted. The availability of **per/pr-grad** is restricted to specified-mass-flow periodic flow calculations while **per/bulk-temp-ratio** is available only for specified-mass-flow periodic heat transfer calculations. Similarly, **time** and **delta_time** are available only if you are modeling unsteady flow.

Window

sets the window ID for the plot of the first statistic selected. The remaining selected statistics will be plotted in windows with incrementally higher IDs.

Axes...

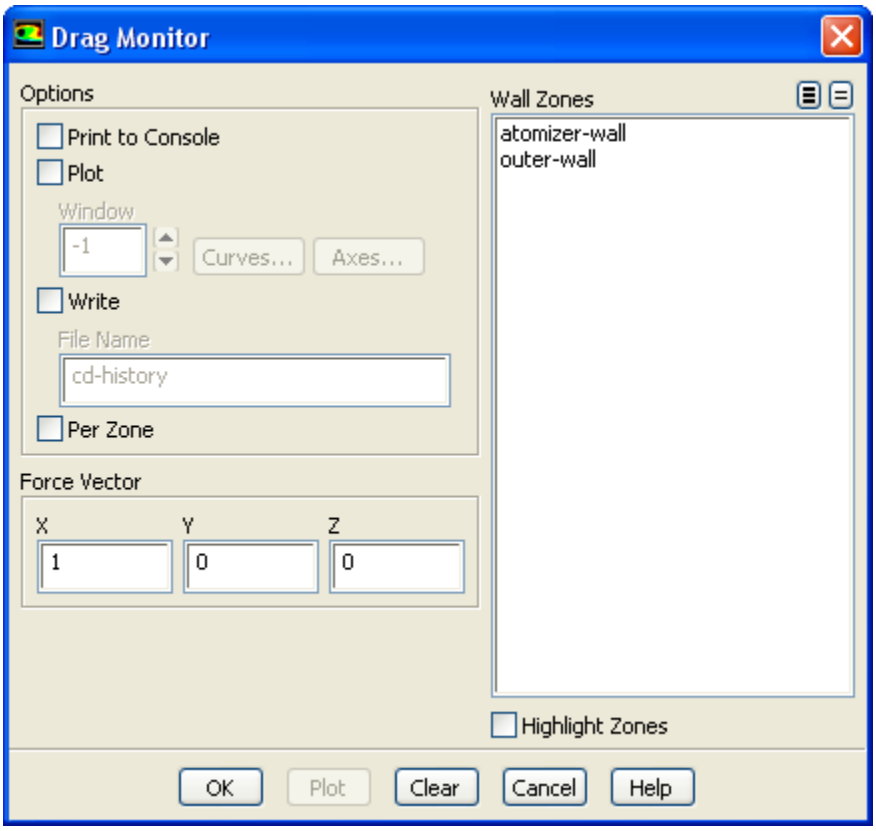
opens the *Axes Dialog Box* (p. 2022) to modify the attributes of the axes.

Curves...

opens the *Curves Dialog Box* (p. 2024) to modify the attributes of the data curves.

35.14.3. Drag Monitor Dialog Box

You can use the **Drag Monitor** dialog box to save the convergence history of the drag coefficient on specified wall zones. See *Monitoring Force and Moment Coefficients* (p. 1278) for details about using this dialog box.



Controls

Options

selects one of the following methods for reporting the selected coefficient:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Print to Console
specifies whether or not to print the selected coefficient in the console window after each iteration.

Plot
specifies whether or not to plot the selected coefficient in the graphics window (with the ID set in **Window**) after each iteration.

Window
sets the window ID in which the plot will be drawn. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the force monitoring plots, and then returned to its previous value. Thus, the force coefficient plots can be maintained in separate windows that do not interfere with other graphical postprocessing.

Axes...
opens the *Axes Dialog Box* (p. 2022) to modify the attributes of the axes.

Curves...
opens the *Curves Dialog Box* (p. 2024) to modify the attributes of the data curves.

Write
specifies whether or not to write the selected coefficient data to a file (with the name in **File Name**) after each iteration.

Important

If you choose *not* to save the force coefficient data in a file, this information will be lost when you exit the current ANSYS FLUENT session.

File Name
specifies the name of the file to which the force data is written (if you are using the **Write** option).

Per Zone
specifies whether or not the forces or moments for multiple walls should be monitored on each wall separately. When this option is turned off (the default), ANSYS FLUENT will compute and monitor the total force for all of the selected walls combined together.

Force Vector
contains the components of the force vector.

X,Y,Z
are the components of the force vector along which the forces will be computed.

Wall Zones
contains a selectable list of wall zones on which the selected coefficient is computed. If you are monitoring more than one coefficient, the list of selected wall zones is often the same for each coefficient. If you want, however, you can have each coefficient computed on a different list of zones.

Highlight Zones
when enabled highlights the zone (selected in the **Drag Monitor** dialog box) in the graphics window.

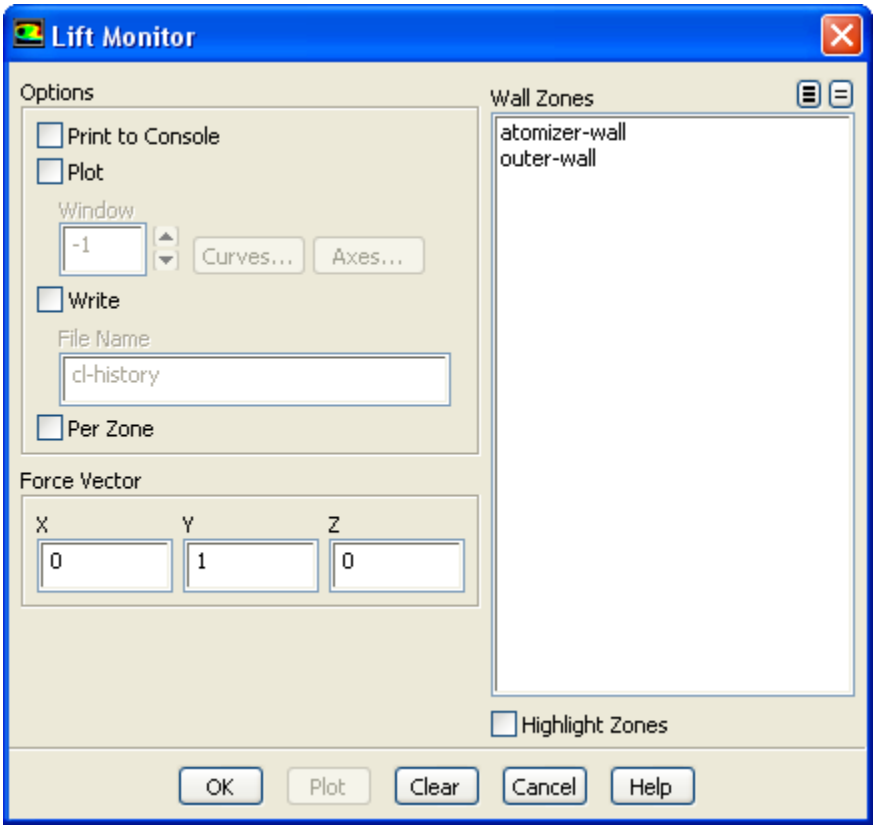
Plot
displays an XY plot of the convergence history of the selected force.

Clear
discards the force-monitoring data, including associated files. Each of these actions must be confirmed in a *Question Dialog Box* (p. 32).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.14.4. Lift Monitor Dialog Box

You can use the **Lift Monitor** dialog box to save the convergence history of the lift coefficient on specified wall zones. See *Monitoring Force and Moment Coefficients* (p. 1278) for details about using this dialog box.



Controls

Options

selects one of the following methods for reporting the selected coefficient:

Print to Console

specifies whether or not to print the selected coefficient in the console window after each iteration.

Plot

specifies whether or not to plot the selected coefficient in the graphics window (with the ID set in **Window**) after each iteration.

Window

sets the window ID in which the plot will be drawn. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the force monitoring plots, and then returned to its previous value. Thus, the force coefficient plots can be maintained in separate windows that do not interfere with other graphical postprocessing.

Axes...

opens the *Axes Dialog Box* (p. 2022) to modify the attributes of the axes.

Curves...

opens the *Curves Dialog Box* (p. 2024) to modify the attributes of the data curves.

Write

specifies whether or not to write the selected coefficient data to a file (with the name in **File Name**) after each iteration.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you choose *not* to save the force coefficient data in a file, this information will be lost when you exit the current ANSYS FLUENT session.

File Name
specifies the name of the file to which the force data is written (if you are using the **Write** option).

Per Zone
specifies whether or not the forces or moments for multiple walls should be monitored on each wall separately. When this option is turned off (the default), ANSYS FLUENT will compute and monitor the total force for all of the selected walls combined together.

Force Vector
contains the components of the force vector.

X,Y,Z
are the components of the force vector along which the forces will be computed.

Wall Zones
contains a selectable list of wall zones on which the selected coefficient is computed. If you are monitoring more than one coefficient, the list of selected wall zones is often the same for each coefficient. If you want, however, you can have each coefficient computed on a different list of zones.

Highlight Zones
when enabled highlights the zone (selected in the **Lift Monitor** dialog box) in the graphics window.

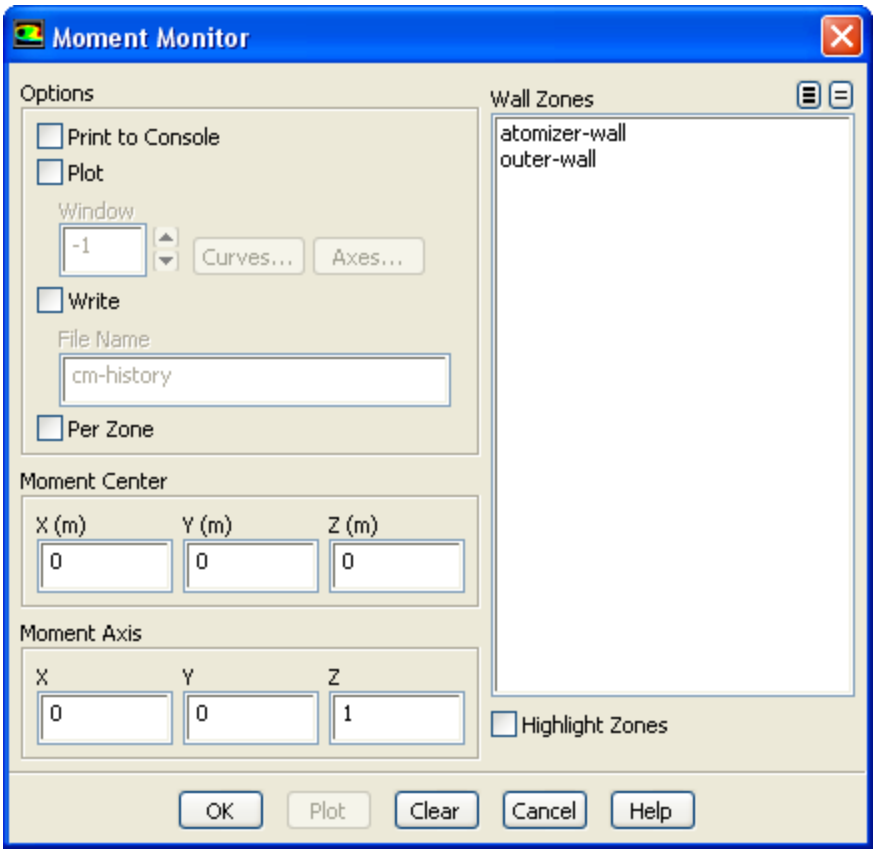
Plot
displays an XY plot of the convergence history of the selected force.

Clear
discards the force-monitoring data, including associated files. Each of these actions must be confirmed in a *Question Dialog Box* (p. 32).

35.14.5. Moment Monitor Dialog Box

You can use the **Moment Monitor** dialog box to save the convergence history of the moment coefficient on specified wall zones. See *Monitoring Force and Moment Coefficients* (p. 1278) for details about using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

selects one of the following methods for reporting the selected coefficient:

Print to Console

specifies whether or not to print the selected coefficient in the console window after each iteration.

Plot

specifies whether or not to plot the selected coefficient in the graphics window (with the ID set in **Window**) after each iteration.

Window

sets the window ID in which the plot will be displayed. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the force monitoring plots, and then returned to its previous value. Thus, the force coefficient plots can be maintained in separate windows that do not interfere with other graphical postprocessing.

Axes...

opens the *Axes Dialog Box* (p. 2022) to modify the attributes of the axes.

Curves...

opens the *Curves Dialog Box* (p. 2024) to modify the attributes of the data curves.

Write

specifies whether or not to write the selected coefficient data to a file (with the name in **File Name**) after each iteration.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you choose *not* to save the force coefficient data in a file, this information will be lost when you exit the current ANSYS FLUENT session.

File Name
specifies the name of the file to which the force data is written (if you are using the **Write** option).

Per Zone
specifies whether or not the moments for multiple walls should be monitored on each wall separately. When this option is turned off (the default), ANSYS FLUENT will compute and monitor the total moment for all of the selected walls combined together.

Moment Center
contains the Cartesian coordinates of the moment center.

X,Y,Z
are the Cartesian coordinates of the moment center about which moments will be computed.

Moment Axis
contains the Cartesian coordinates of the moment vector to be monitored. For two-dimensional flows, only the moment vector about the z-coordinate axis exists. Presently, you can monitor only one component of the moment vector at a time.

Wall Zones
contains a selectable list of wall zones on which the selected coefficient is computed. If you are monitoring more than one coefficient, the list of selected wall zones is often the same for each coefficient. If you want, however, you can have each coefficient computed on a different list of zones.

Highlight Zones
when enabled highlights the zone (selected in the **Moment Monitor** dialog box) in the graphics window.

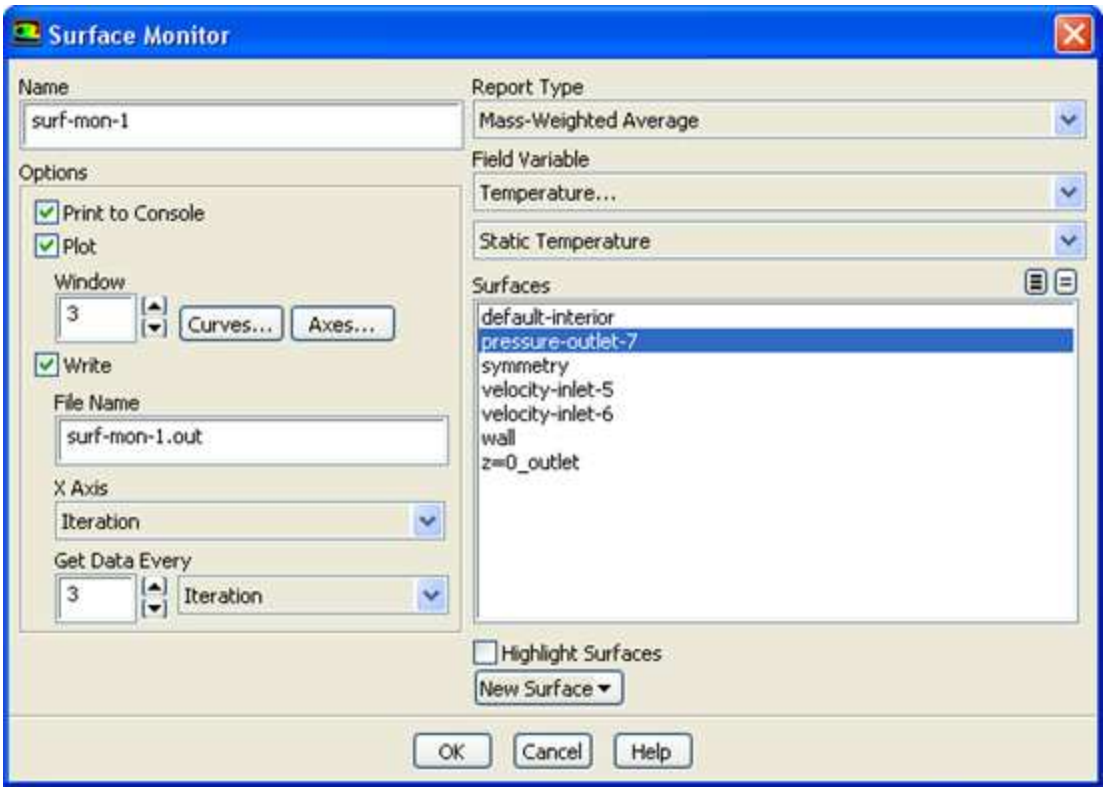
Plot
displays an XY plot of the convergence history of the selected moment coefficient.

Clear
discards the force-monitoring data, including associated files. Each of these actions must be confirmed in a *Question Dialog Box* (p. 32).

35.14.6. Surface Monitor Dialog Box

The **Surface Monitor** dialog box allows you to save the convergence history of the average, integral, flow rate, or mass average (among other quantities) of a field variable on one or more surfaces. You can also use the **Surface Monitor** dialog box define what each surface monitor is tracking. See *Monitoring Surface Integrals* (p. 1283) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the surface monitor.

Options

contains parameters used in saving monitor data.

Print to Console

specifies whether or not to print the data from each surface monitor in the console window.

Plot

specifies whether or not to plot the data from each surface monitor in the graphics window.

Window

sets the window ID in which the plot will be drawn. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the surface monitoring plot, and then returned to its previous value. Thus, the surface monitor plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

Curves...

opens the *Curves Dialog Box* (p. 2024) to modify the attributes of the data curves.

Axes...

opens the *Axes Dialog Box* (p. 2022) to modify the attributes of the axes.

Write

specifies whether or not to write the data from each surface monitor to a file.

Important

If you choose *not* to save the surface integration data in a file, this information will be lost when you exit the current ANSYS FLUENT session.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Name
specifies the name of the file to which the surface data is written.

X Axis
specifies the x-axis function against which monitored data will be plotted or written. Available options are **Iteration**, **Time Step**, and **Flow Time** (elapsed time).

Get Data Every
indicates the frequency at which you want to plot, print, or write the surface monitor. A default value of 1 will allow you to monitor at every **Iteration** or **Time Step**. **Time Step** is a valid choice only if you are calculating unsteady flow. See *Overview of Defining Surface Monitors* (p. 1283) for details.

Report Type
selects the integration method used on the selected surfaces. The available report types are the same as those in the *Surface Integrals Dialog Box* (p. 2031). See *Surface Integration* (p. 1519) for details.

Field Variable
contains a list of solution variables that can be monitored on the selected surfaces. This list will be deactivated if you select **Mass Flow Rate** or **Volume Flow Rate** as the **Report Type**.

Phase
contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Surfaces
contains a selectable list of the current surfaces.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the *Point Surface Dialog Box* (p. 1927).

Line/Rake
opens the *Line/Rake Surface Dialog Box* (p. 1928).

Plane
opens the *Plane Surface Dialog Box* (p. 1929).

Quadric
opens the *Quadric Surface Dialog Box* (p. 1931).

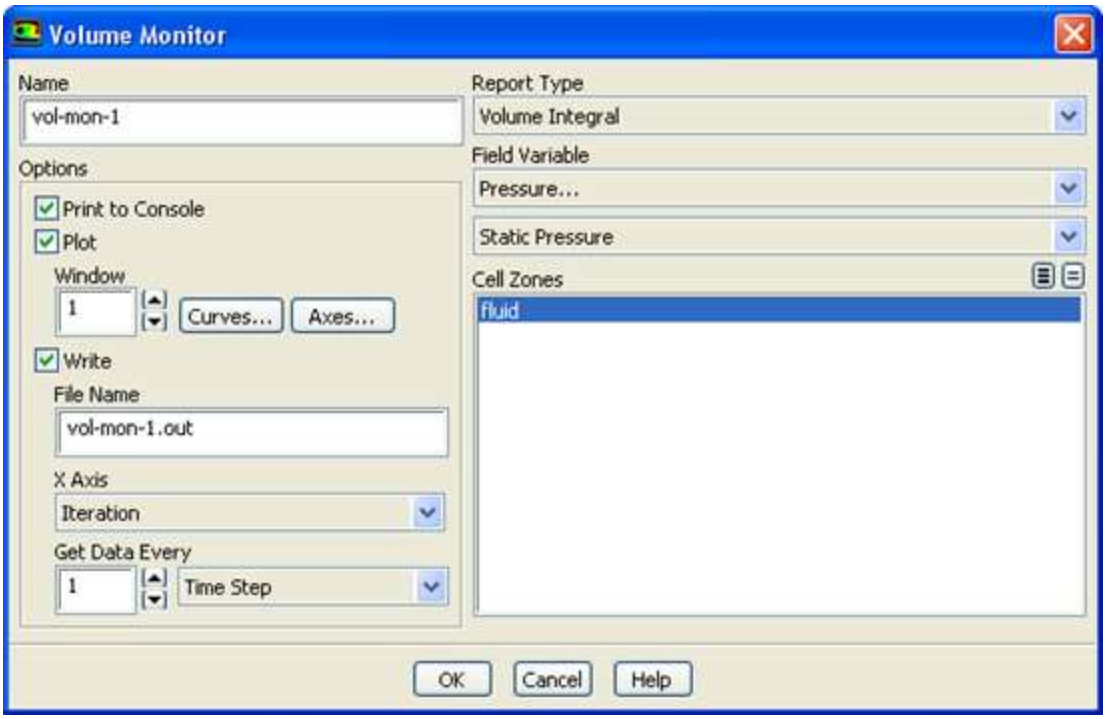
Iso-Surface
opens the *Iso-Surface Dialog Box* (p. 1933).

Iso-Clip
opens the *Iso-Clip Dialog Box* (p. 1935).

35.14.7. Volume Monitor Dialog Box

You can use the **Volume Monitor** dialog box to save the convergence history of the volume, sum, volume integral, mass integral, volume average, or mass average of a field variable on one or more cell zones. You can also define what each volume monitor is tracking. See *Monitoring Volume Integrals* (p. 1286) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the volume monitor.

Options

contains parameters used in saving monitor data.

Print to Console

specifies whether or not to print the data from each volume monitor in the console window.

Plot

specifies whether or not to plot the data from each volume monitor in the graphics window.

Window

sets the window ID in which the plot will be drawn. When ANSYS FLUENT is iterating, the active graphics window is temporarily set to this window to update the volume monitoring plot, and then returned to its previous value. Thus, the volume monitor plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

Curves...

opens the *Curves Dialog Box* (p. 2024) to modify the attributes of the data curves.

Axes...

opens the *Axes Dialog Box* (p. 2022) to modify the attributes of the axes.

Write

specifies whether or not to write the data from each volume monitor to a file.

Important

If you choose *not* to save the volume integration data in a file, this information will be lost when you exit the current ANSYS FLUENT session.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Name
specifies the name of the file to which the volume monitor data is written.

X Axis
specifies the x-axis function against which monitored data will be plotted or written. Available options are **Iteration**, **Time Step**, and **Flow Time** (elapsed time).

Get Data Every
indicates the frequency at which you want to plot, print, or write the volume monitor. A default value of 1 will allow you to monitor at every **Iteration** or **Time Step**. **Time Step** is a valid choice only if you are calculating unsteady flow. See *Overview of Defining Surface Monitors* (p. 1283) for details.

Report Type
selects the integration method used on the selected cell zones. The available report types are the same as those in the *Volume Integrals Dialog Box* (p. 2034). See *Volume Integration* (p. 1521) for details.

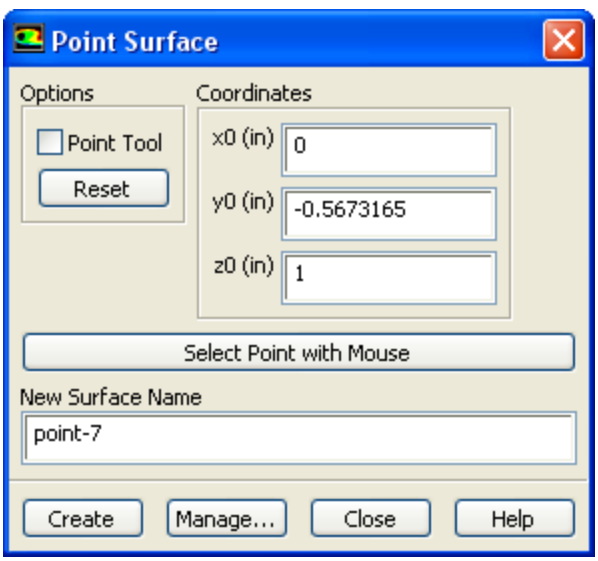
Field Variable
contains a list of solution variables that can be monitored on the selected cell zones. This list will be deactivated if you select **Volume** as the **Report Type**.

Phase
contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Cell Zones
contains a selectable list of the current cell zones.

35.14.8. Point Surface Dialog Box

The **Point Surface** dialog box allows you to interactively create point surfaces (surfaces containing a single data point). See *Point Surfaces* (p. 1359) for details about the items below.



Controls

Options
contains options related to the point tool. See *Using the Point Tool* (p. 1360) for details about using this feature.

Point Tool
activates the point tool.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset
resets the point tool to its default position.

Coordinates
designates the coordinates of the point in the surface (**x0**, **y0**, **z0**).

Select Point with Mouse
activates the selection of the point with the mouse. You can select a point by clicking on a location in the active window with the mouse-probe button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about setting mouse button functions.)

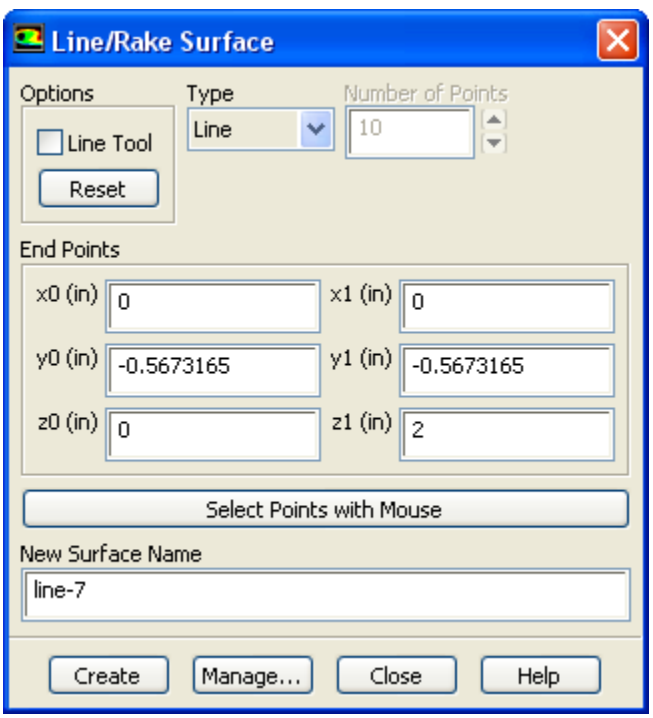
New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

35.14.9. Line/Rake Surface Dialog Box

The **Line/Rake Surface** dialog box allows you to interactively create line and rake surfaces. A rake is a linear, uniform distribution of points between two endpoints. See *Line and Rake Surfaces* (p. 1361) for details about the items below.



Controls

Options
contains options related to the line tool. See *Using the Line Tool* (p. 1363) for details about using this feature.

Line Tool
activates the line tool.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset
resets the line tool to its default position.

Type
selects line or rake as the surface to be created.

Number of Points
defines the number of points in the rake surface (inactive for line surfaces).

End Points
designates the coordinates of the first point (**x0**, **y0**, **z0**) and the last point (**x1**, **y1**, **z1**).

Select Points with Mouse
activates the selection of endpoints with the mouse. You can select endpoints by clicking on locations in the active window with the mouse-probe button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about setting mouse button functions.)

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

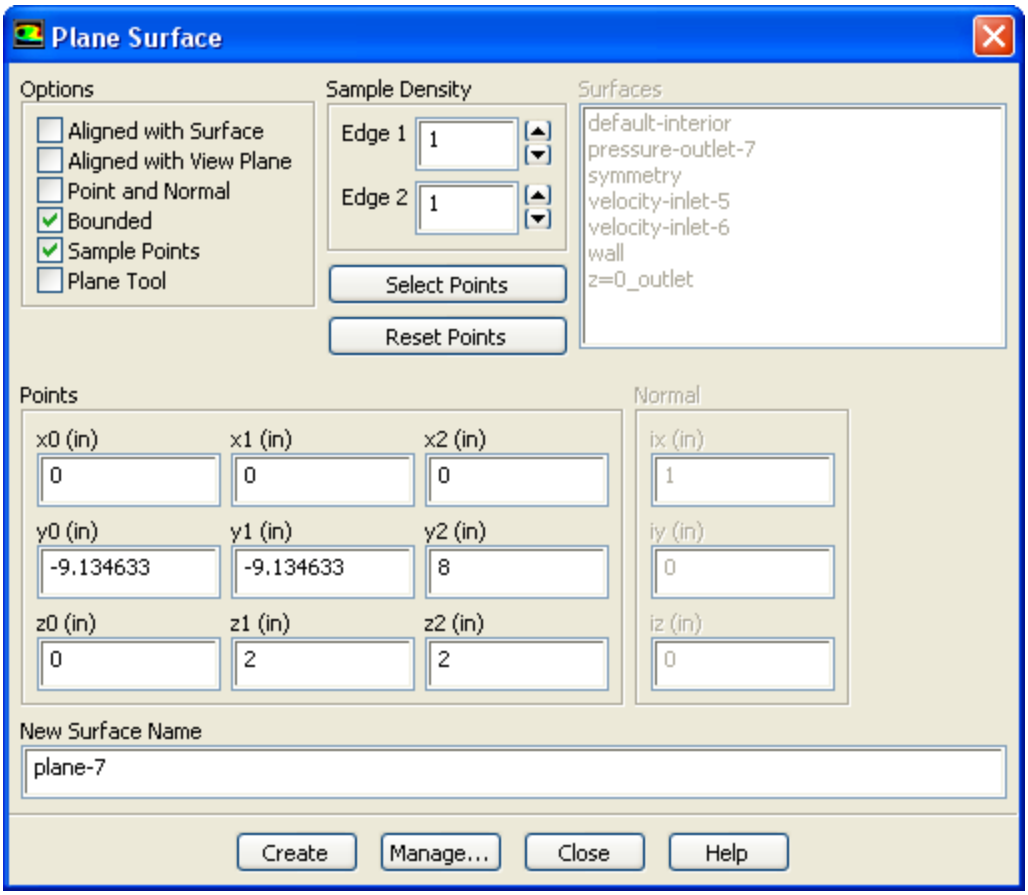
Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

35.14.10. Plane Surface Dialog Box

The **Plane Surface** dialog box allows you to interactively create a planar surface that cuts through the domain. See *Plane Surfaces* (p. 1364) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains options for defining the plane surface.

Aligned with Surface

enables the specification of a plane parallel to an existing surface.

Aligned with View Plane

enables the specification of a plane parallel to the current view in the active graphics window.

Point and Normal

enables the specification of a plane having a certain normal vector and passing through a specified point.

Bounded

enables the creation of a bounded parallelepiped, 3 of whose 4 corners are the 3 points that define the plane equation (or the 4 corners of the **Plane Tool**).

Sample Points

enables the specification of a point density along the 2 directions (of the parallelepiped). This creates a uniformly distributed set of points on the plane. (This item is available only when the **Bounded** option is enabled.)

Plane Tool

activates the plane tool. See *Using the Plane Tool* (p. 1367) for details. This option is not available if you are using the **Aligned With Surface** or **Aligned With View Plane** option.

Sample Density

specifies the density of points when the **Sample Points** option is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edge 1, Edge 2
sets the point density along the two directions of the plane. (Edge 1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2.)

Select Points
activates the selection of points with the mouse. You can select endpoints by clicking on locations in the active window with the mouse-probe button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about setting mouse button functions.)

Reset Points
resets the plane tool to its default position.

Surfaces
contains a list of currently defined surfaces. If you choose the **Aligned With Surface** option, this will become a selectable list, and you can choose the surface with which you want the new plane surface to be aligned.

Points
contains boxes in which you can set the coordinates of the three points defining the planar surface.

x0, y0, z0
designates the coordinates of the first point.

x1, y1, z1
designates the coordinates of the second point.

x2, y2, z2
designates the coordinates of the third point.

Normal
contains boxes in which you can specify the components of the normal vector when the **Point And Normal** option is enabled.

ix, iy, iz
designates the coordinates of the end point of the normal vector (the start point being 0, 0, 0).

New Surface Name
designates the name of the new surface The default is the concatenation of the surface type and an integer which is the new surface ID.

Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

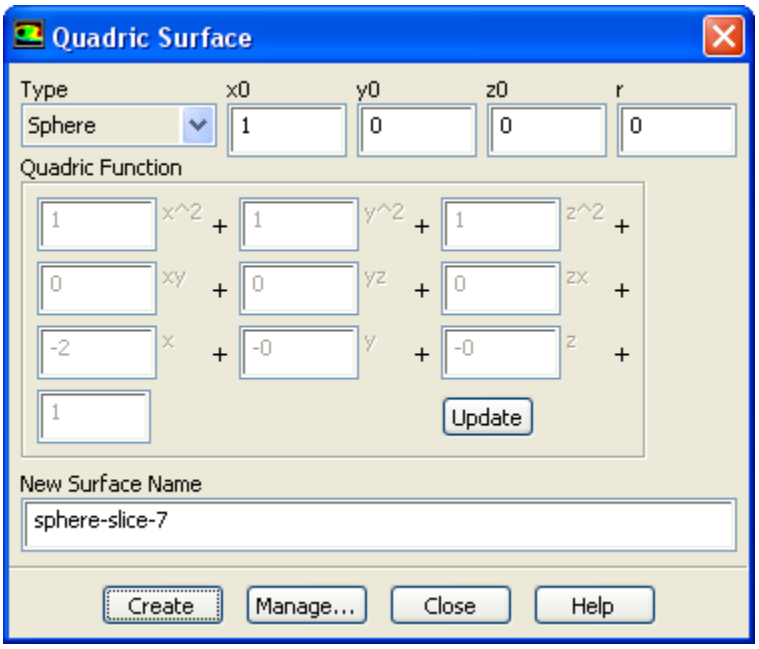
35.14.11. Quadric Surface Dialog Box

The **Quadric Surface** dialog box allows you to interactively define quadric functions and create surfaces from them. Lines and circles are commonly used types of quadric functions, so additional support is provided for defining them. See *Quadric Surfaces* (p. 1369) for details about using the items below.

Important

Note that your inputs for the quadric function must be in SI units.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Type

selects the type of quadric function. Presently, you may select **Line/Plane**, **Circle/Sphere** or a general **Quadric**. Some of the control buttons take on different meanings for each of the three quadric function types.

Line/Plane

specifies that the surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=distance$, where

ix

is the coefficient of x in the quadric function.

iy

is the coefficient of y in the quadric function.

iz

is the coefficient of z in the quadric function.

distance

is the distance of the line/plane from the origin.

When you press the **Update** button in **Quadric Function**, the coefficients in the quadric function will change to reflect your inputs.

Circle/Sphere

specifies that the surface will consist of all points in the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$, where

x0,y0,z0

are the x,y,z coordinates of the center.

r

is the radius.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you press the **Update** button in **Quadric Function**, the coefficients in the quadric function will change to reflect your inputs.

Quadric
specifies that the surface will consist of all points in the domain that satisfy the quadric function $Q = value$, where Q is the quadric function defined by the user as shown below. You will enter **value** to the right of the **Type** entry, but the coefficients in the quadric function are entered directly in the **Quadric Function** box. Note that in 2D problems the z entries are ignored.

x2, y2, z2, xy, yz, zx, x, y, z
are the coefficients of the corresponding terms in the quadric function.

Quadric Function
contains the display of the quadric function and the **Update** button. If you select **Line/Plane** or **Circle/Sphere** as the **Type**, the coefficients of the quadric function will be updated here when you press the **Update** button. You will not be able to edit the values in the **Quadric Function** box directly. If you select **Quadric** as the **Type**, you will enter the coefficients directly in this box, and the **Update** button will be inactive.

Update
updates the coefficients in **Quadric Function** to reflect the current input.

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

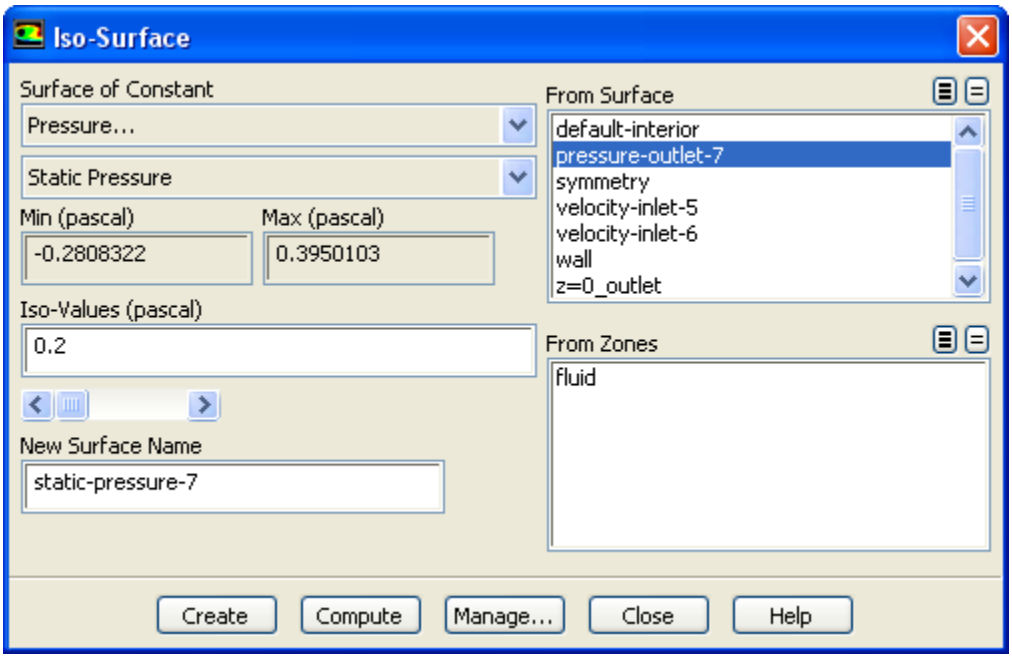
Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

35.14.12. Iso-Surface Dialog Box

The **Iso-Surface** dialog box allows you to interactively create isosurfaces. These surfaces can be isovalued sections of an existing surface or of the entire domain. For more effective use of the slider bar, press the **Compute** button before using it. See *Isosurfaces* (p. 1371) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Surface of Constant

contains a list from which you can select the scalar field which will be used for isosurfacing.

Min

displays the minimum field value, which is computed when you press **Compute**.

Max

displays the maximum field value, which is computed when you press **Compute**.

Iso-Values

sets user-specified isovalues. There are two ways you can set the **Iso-Values**:

- You can set an isovalue interactively by moving the slider with the left mouse button. This will also create a temporary isosurface in the graphics window. Using the slider allows you to preview the isosurfaces before defining them. Note: Even though the isosurface is displayed, it is only a temporary surface. To create an isosurface, use the **Create** button after deciding on a particular isovalue.
- You can type in isovalues in the **Iso-Values** field directly, separating multiple values by white space. Multiple isovalues will be contained in a single isosurface; i.e., you cannot select subsurfaces within the resulting isosurface.

From Surface

contains a list of existing surfaces from which you can select the surface to be used for isosurfacing. If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.

From Zones

contains a list of cell zones from which you can select the zone for creating an isosurface.

New Surface Name

designates the name of the surface to be created. The default is the concatenation of the scalar field name and an integer which is the new surface ID.

Create

creates the surface.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute

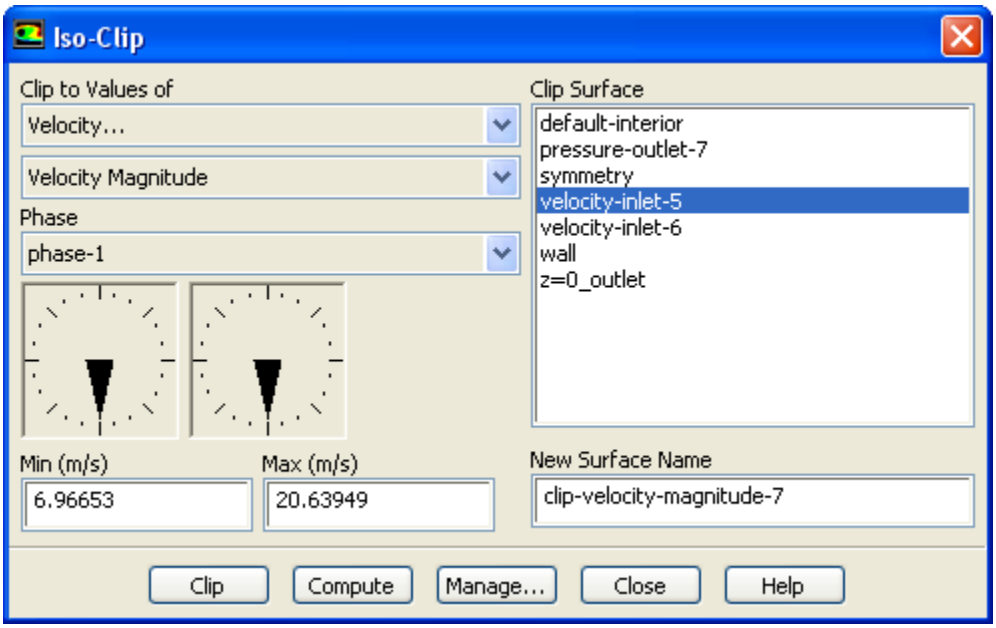
computes the minimum and maximum of the scalar field across the domain and displays them in the **Min** and **Max** boxes.

Manage...

opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

35.14.13. Iso-Clip Dialog Box

The **Iso-Clip** dialog box allows you to interactively clip surfaces. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. See *Clipping Surfaces* (p. 1373) for details about the items below.



Controls

Clip to Values of

contains a list from which you can select the scalar field to be used for clipping.

Phase

allows you to select the phase when one of the multiphase models is selected in the *Multiphase Model Dialog Box* (p. 1642).

Min, Max

set the minimum and maximum values in the clipping range. There are two ways you can set the **Min** and **Max**:

- You can set a value interactively by moving the indicator in the dial above the **Min** or **Max** field with the left mouse button. This will also create a temporary surface in the graphics window. Using the dial allows you to preview the clipped surfaces before defining them. Note: Even though the clipped surface is displayed, it is only a temporary surface. To clip the surface, use the **Clip** button after deciding on the minimum and maximum values.
- You can type in values in the **Min** and **Max** fields directly.

Clip Surface

contains a list of surfaces from which you can select the surface to be clipped. You must select a surface to activate the **Clip** push button.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Surface Name
designates the name of the surface to be created. The default is the concatenation of the surface type and an integer which is the new surface ID.

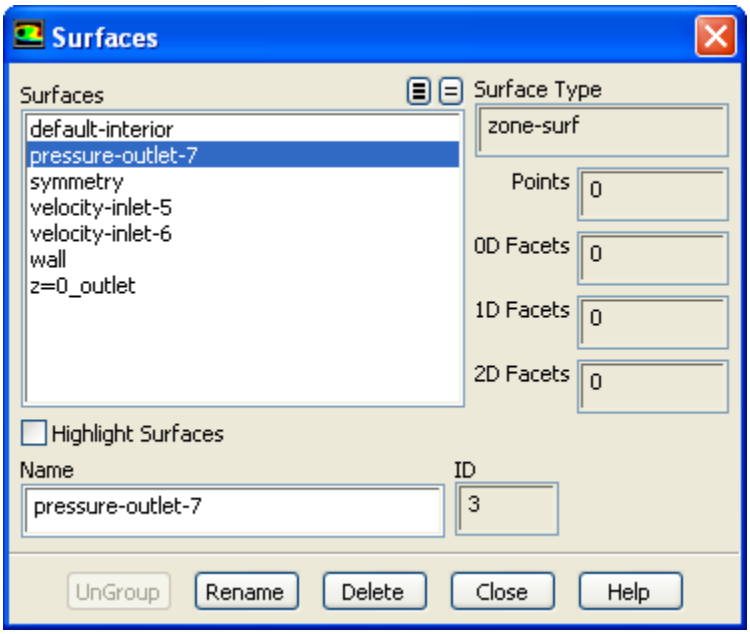
Clip
creates the clipped surface. (The original surface will remain unchanged.)

Compute
computes the minimum and maximum values of the scalar field across the domain, and displays them in the **Min** and **Max** boxes.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

35.14.14. Surfaces Dialog Box

The **Surfaces** dialog box allows you to interactively group, rename, and delete surfaces and obtain information about their components. See *Grouping, Renaming, and Deleting Surfaces* (p. 1376) for details about the items below.



Controls

Surfaces
contains a list of existing surfaces from which you can select the surface(s) of interest.

Name
displays the name of the selected surface. You can edit the text field to modify the surface name. (If more than one surface is selected, the name of the first one you selected will be displayed.)

Surface Type
displays the type of surface that is selected (e.g., `zone-surf` if one surface is selected, or `Multiple Surfaces` if more than one surface is selected).

Points, 0D Facets, 1D Facets,
and **2D Facets** display the number of points and facets in the selected surface. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.

Highlight Surfaces
when enabled highlights the surfaces (selected in the **Surfaces** dialog box) in the graphics window.

ID
displays the ID of the selected surface. You cannot change this value.

UnGroup
ungroups the selected surface. This button is available only if the selected surface was created by **Group**ing two or more surfaces together.

Rename
renames the selected surface in **Surfaces** with the name specified in **Name**. This button is available when just one surface is selected. (If two or more surfaces are selected, it becomes the **Group** button.)

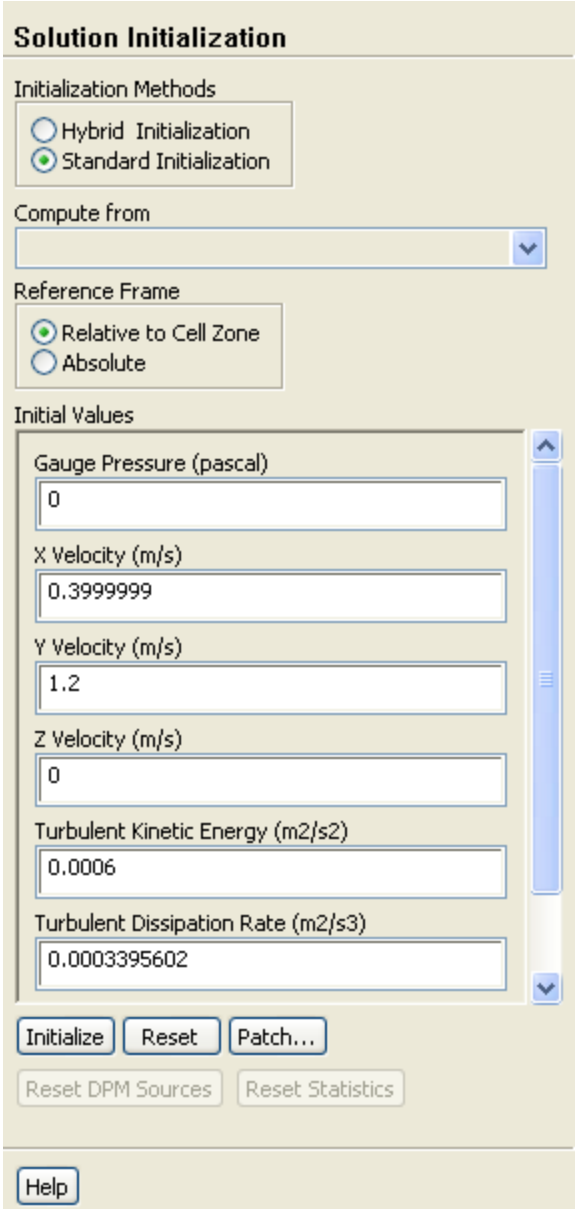
Group
groups two or more selected surfaces and gives the group the name entered in **Name**. This button replaces the **Rename** button when two or more surfaces are selected.

Delete
deletes the selected surface(s).

35.15. Solution Initialization Task Page

The **Solution Initialization** task page allows you to define values for flow variables and initialize the flow field to these values. See *Initializing the Entire Flow Field Using Standard Initialization* (p. 1240) for details about using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Initialization Method

allows you to choose between **Hybrid Initialization** and **Standard Initialization**.

Hybrid Initialization

is a collection of boundary interpolation methods, where variables, such as temperature, turbulence, species fractions, volume fractions, etc., are automatically patched based on domain averaged values or a particular interpolation recipe (see *Hybrid Initialization* (p. 1245)).

Standard Initialization

allows you to define values for flow variables and initialize the flow field to these values.

Compute from

is a drop-down list of zones; the default values for applicable variables will be computed from information contained in the zone that you select from this list. The computation will occur when you select the required zone, and the variable values will be displayed in **Initial Values**. You can also choose the **all-zones** item in this list to compute average values based on all zones.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Frame
indicates whether the initial velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Initial Values
displays the initial values of applicable variables. You can use **Compute from** to compute values from a particular zone, or you can enter values directly.

Initialize
initializes the entire flow field to the values listed.

Reset
resets the fields to their "saved" values.

Patch...
opens the *Patch Dialog Box* (p. 1939).

Reset DPM Sources
allows you to reset the interphase sources/sinks of momentum, heat, and/or mass to zero. This item is available when you perform coupled discrete phase calculations. See *Resetting the Interphase Exchange Terms* (p. 1067) for details.

Reset Statistics
can be used to both initialize the flow statistics and reset the flow statistics after you have gathered some data for time statistics. This item is available when you perform unsteady calculations and have enabled the **Data Sampling for Time Statistics** option in the *Run Calculation Task Page* (p. 1955). See *User Inputs for Time-Dependent Problems* (p. 1256)for details.

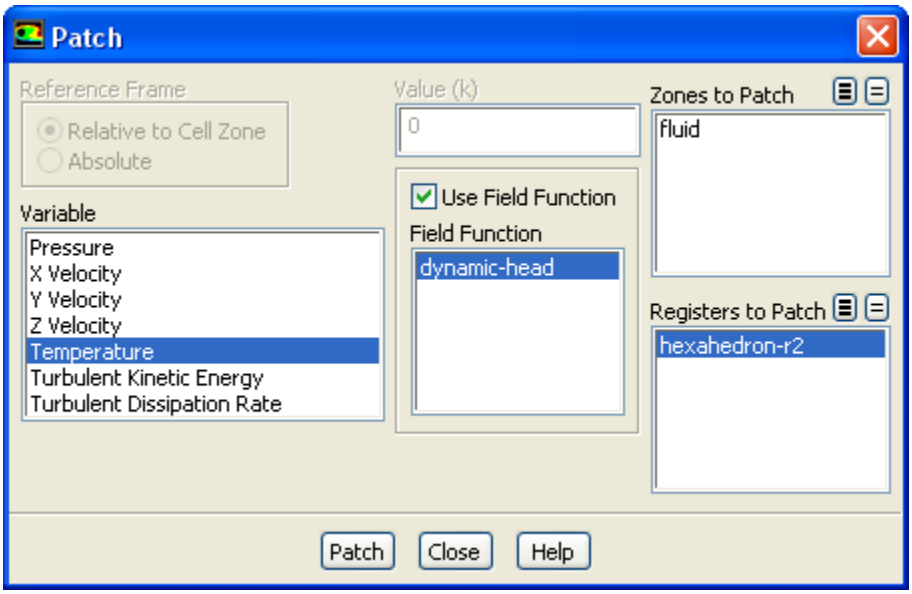
More Settings...
opens the *Hybrid Initialization Dialog Box* (p. 1941). This button is available only when **Hybrid Initialization** is the selected method.

For additional information, please see the following sections:
35.15.1. Patch Dialog Box
35.15.2. Hybrid Initialization Dialog Box

35.15.1. Patch Dialog Box

The **Patch** dialog box allows you to patch different values of flow variables into different cells. See *Patching Values in Selected Cells* (p. 1242) for details about using this feature.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reference Frame

indicates whether patched velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent. Also, this selection affects only velocities, so the options will not be available unless you have selected a velocity in the **Variable** list.

Phase

contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Variable

contains a list of flow variables from which you can choose the variable to be patched.

Value

sets a constant value to be patched for the selected **Variable**.

Use Field Function

enables the patching of a custom **Field Function**, rather than a constant **Value**, for the selected **Variable**. See *Using Field Functions* (p. 1243) for details.

Field Function

contains a list of defined custom field functions. If the **Use Field Function** option is enabled, you can patch one of these functions for the selected **Variable**. See *Using Field Functions* (p. 1243) for details.

Zones to Patch

contains a list of cell zones. The specified **Value** or **Field Function**for the selected **Variable** will be patched into the zones you select from this list.

Registers to Patch

contains a list of cell registers that have been created using the adaption tools. You can patch a different value for a group of cells within a single cell zone by selecting a register containing a subset of the cells in the zone. See *Using Registers* (p. 1243) for details.

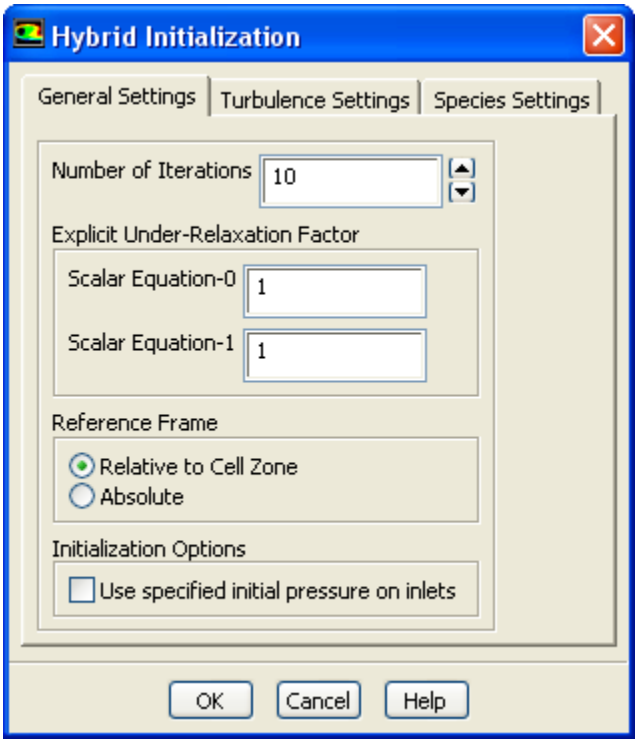
Patch

updates the flow-field data based on the inputs above.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.15.2. Hybrid Initialization Dialog Box

The **Hybrid Initialization** dialog box contains a host of adjustable settings which control the **Hybrid Initialization** strategy.



Controls

General Settings

allow you to adjust such entries as the number of iterations and under-relaxation factors.

Number of Iterations

uses a default value of 10. This is the number of iterations that will be performed while solving the Laplace equations to initialize the velocity and pressure. If the initialized velocity and pressure fields are not to your liking, you may want to increase the number of iterations and re-initialize the solution.

Explicit Under-Relaxation Factor

uses a default value of 1. This value will be used while solving the Laplace equation to initialize the velocity and pressure. If the initialized velocity and pressure fields are not to your liking, you may want to adjust the under-relaxation factor and re-initialize the solution.

Reference Frame

indicates whether the initial velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Initialization Option

can be enabled if you want that the specified pressure for **Supersonic/Initialization Gauge Pressure** at the inlet boundaries to be used.

Turbulence Settings

uses by default the domain averaged values for the turbulence parameters. If you want to use variable turbulence parameters you can deselect the **Average Turbulent Parameters** check box. When this option

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

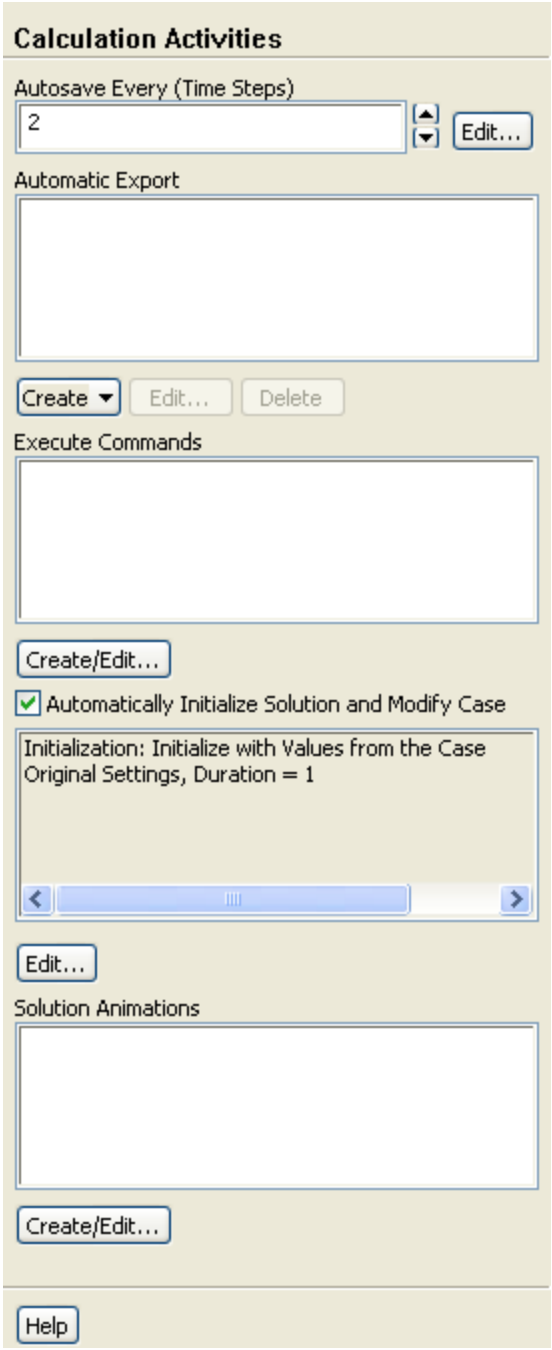
is disabled, then it calculates the turbulent parameters, such as kinetic energy, dissipation energy, etc., using local flow parameters.

Species Settings

will by default initialize secondary species with zero mass or mole fractions. If you want to specify the appropriate value for the species, you will need to enable **Specify Species Parameters**.

35.16. Calculation Activities Task Page

The **Calculation Activities** task page allows you to set up various tasks that you can perform during the calculation, such as saving files, exporting files, creating solution animations, and command execution.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Autosave Every
allows you to set the frequency of the autosave.

Edit...
opens the *Autosave Dialog Box* (p. 1943).

Automatic Export
displays a list of available export objects that will be executed during the calculations. This list and its associated buttons are only available for transient flow calculations.

Create
provides a drop-down list that contains options for creating export objects. Two options are available: the **Solution Data Export** option opens the *Automatic Export Dialog Box* (p. 1946); the **Particle History Data Export** option opens the *Automatic Particle History Data Export Dialog Box* (p. 1949).

Edit...
opens the appropriate dialog box for the selected item in the **Automatic Export** list.

Delete
removes the selected item from the **Automatic Export** list.

Execute Commands
lists available commands to be executed during the calculations.

Create/Edit...
opens the *Execute Commands Dialog Box* (p. 1950).

Automatically Initialize Solution and Modify Case
allows you to automatically have your solution initialized and your case file modified.

Edit...
opens the *Automatic Solution Initialization and Case Modification Dialog Box* (p. 1952). This button is only available when the **Automatically Initialize Solution and Modify Case** option is enabled.

Solution Animations
lists available solution animations.

Create/Edit...
opens the *Solution Animation Dialog Box* (p. 1953).

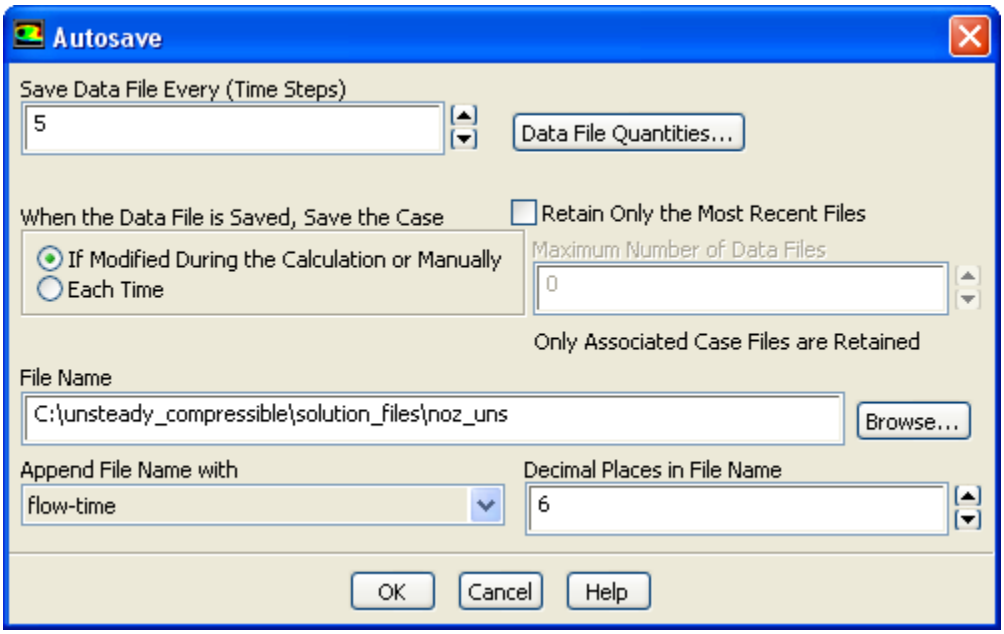
For additional information, please see the following sections:

35.16.1. Autosave Dialog Box
35.16.2. Data File Quantities Dialog Box
35.16.3. Automatic Export Dialog Box
35.16.4. Automatic Particle History Data Export Dialog Box
35.16.5. Execute Commands Dialog Box
35.16.6. Define Macro Dialog Box
35.16.7. Automatic Solution Initialization and Case Modification Dialog Box
35.16.8. Solution Animation Dialog Box
35.16.9. Animation Sequence Dialog Box

35.16.1. Autosave Dialog Box

The **Autosave** dialog box allows you to specify automatic saving of case and data files at specified intervals during a calculation. See *Automatic Saving of Case and Data Files* (p. 65) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Save Data File Every

specifies the frequency with which data files are saved. For steady flows you will specify the frequency in iterations, while for unsteady flows you will specify it in time steps (unless you are using the explicit time stepping formulation, in which case you will specify the frequency in iterations). The default value is set to zero, indicating that no automatic saving is performed.

When the Data File is saved, Save the Case

gives you the choice to save the case file only if it is modified or each time that the data file is saved.

If Modified During the Calculation or Manually

will result in ANSYS FLUENT saving a case file whether you make a manual change, or if ANSYS FLUENT makes a change to the code internally during the calculation.

Each Time

allows you to save the case file every time the data file is saved.

Retain Only the Most Recent Files

allows you to restrict the number of files saved by ANSYS FLUENT if you have limited disk space. When this option is enabled, you can enter the appropriate value in the **Maximum Number of Data Files** field.

Only the associated case files are retained when using this option.

Data File Quantities

opens the *Data File Quantities Dialog Box* (p. 1945) where you can specify which quantities you want to automatically save to a data file for postprocessing.

Maximum Number of Data Files

specifies the maximum number of data files that can be saved at any instance. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, ANSYS FLUENT will overwrite the earliest existing file. The default value for this field is zero which saves all the files.

File Name

specifies the root name for the files that are saved. The iteration or time-step number and an appropriate suffix (`.cas` or `.dat`) will be added to the specified root name. If the specified **File Name** ends in `.gz`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or .Z, appropriate file compression will be performed. (See *Reading and Writing Compressed Files* (p. 59) for details about file compression.)

Append File Name with

allows you to select **flow-time** or **time-step** to be appended to the file name. This option is available only for unsteady-state calculations. The default selection is **flow-time**.

Decimal Places in File Name

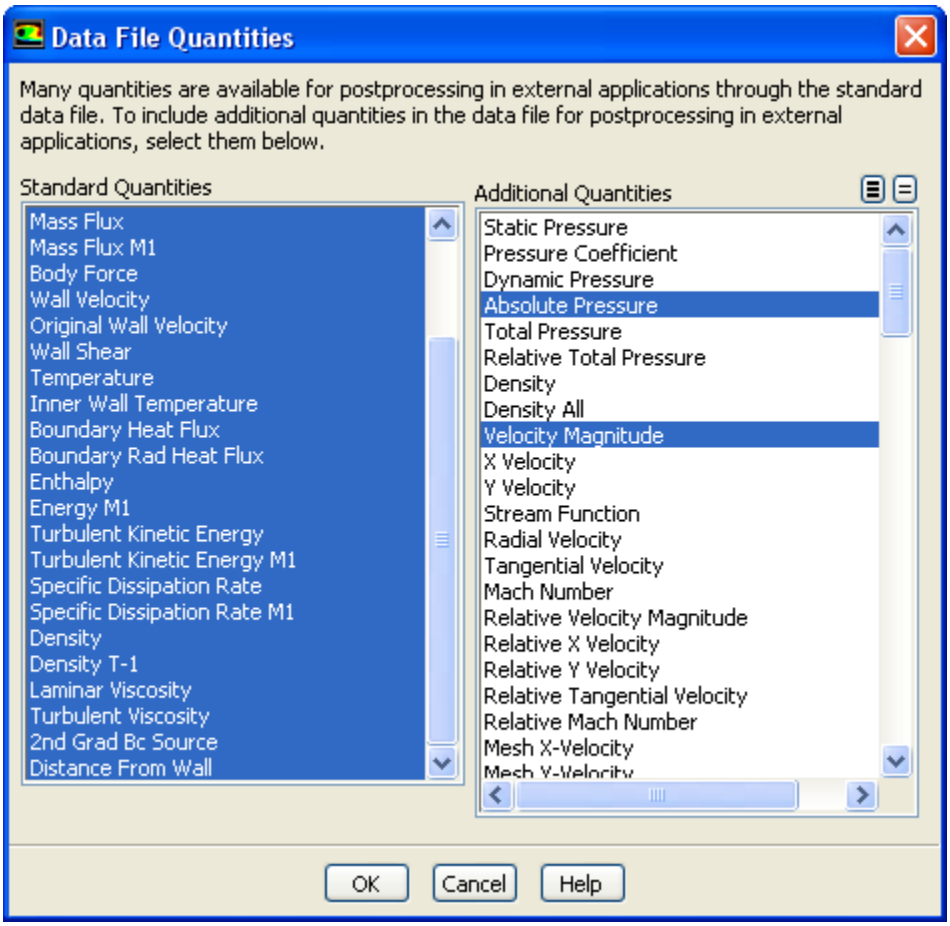
allows you to specify the number of decimal digits in the file name. This option is available only when **flow-time** is selected in the **Append File Name with** drop-down list. The default value for this field is set to 6.

Important

The **Autosave** dialog box is slightly different when running ANSYS FLUENT within ANSYS Workbench. For more information, see the separate ANSYS FLUENT in **Workbench** User's Guide.

35.16.2. Data File Quantities Dialog Box

The **Data File Quantities** dialog box allows you to specify various quantities for postprocessing. See *Setting Data File Quantities* (p. 121) for details.



Controls

Standard Quantities

contains a listing of standard postprocessing quantities (e.g., density, Mach number, temperature, etc.).

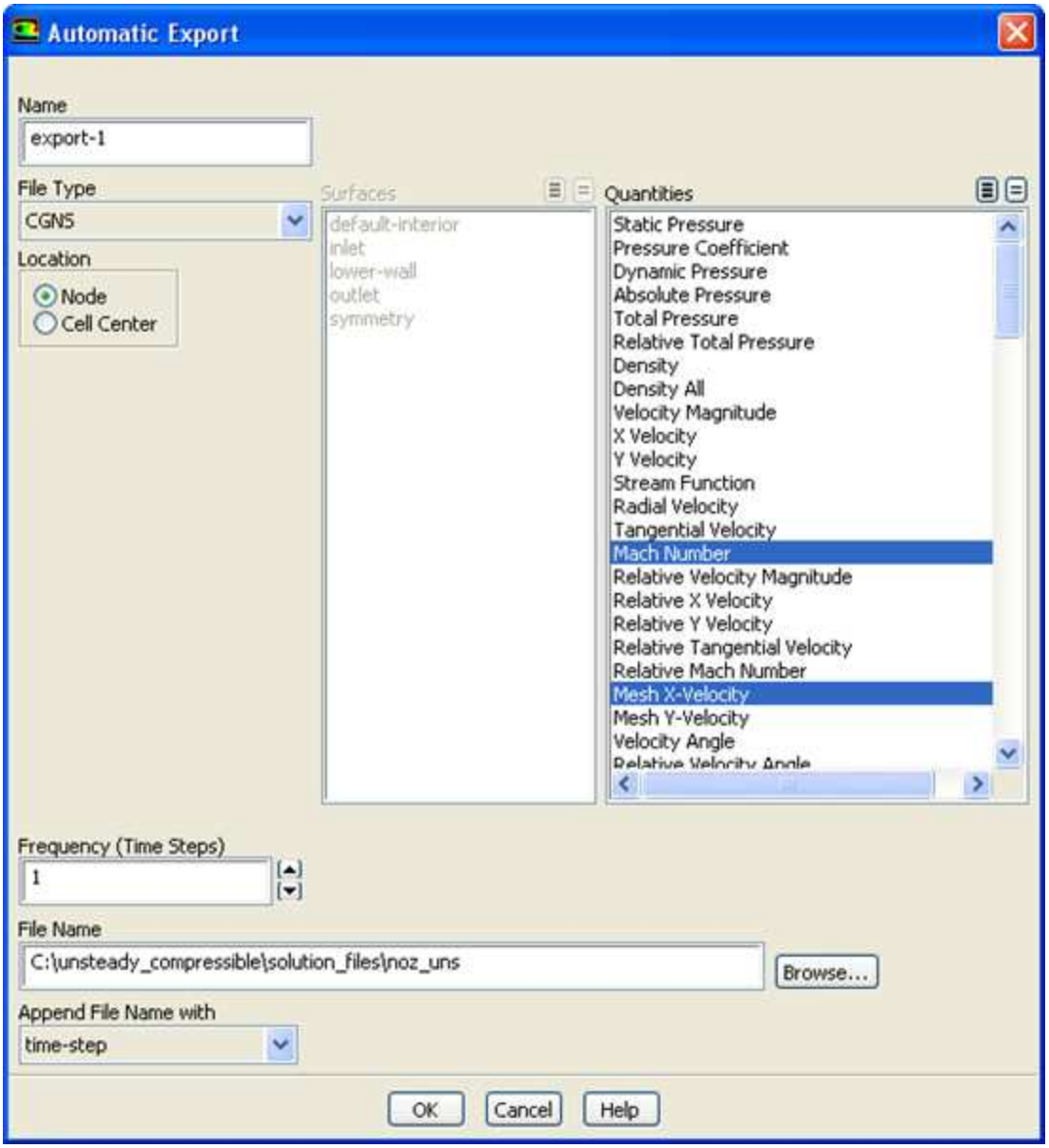
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Quantities

contains a listing of additional postprocessing quantities that are derived from the standard quantities (e.g., standard pressure, velocity magnitude, etc.).

35.16.3. Automatic Export Dialog Box

The **Automatic Export** dialog box allows you to create an automatic export definition for solution data. See *Creating Automatic Export Definitions for Solution Data* (p. 100) for details.



Controls

Name

specifies the name of the export definition.

File Type

contains a drop-down list of file types, which control the output file format that will be written.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ABAQUS
allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to an ABAQUS file (extension `.inp`).

ASCII
allows you to specify the surface(s), scalars, location from which the values of scalar functions are to be taken, and the delimiter separating the fields, to be exported to an **ASCII** file.

AVS
allows you to specify the scalars you want to write to be exported to an AVS file.

CFD-Post Compatible
allows you to specify the scalars you want to write to be exported to a file compatible to the **CFD-Post** application.

CGNS
allows you to specify the scalars you want to write and the location from which the values of scalar functions are to be taken, to be exported to a **CGNS** file (extension `.cgns`).

Data Explorer
allows you to specify the surface(s) and the scalars you want to write to be exported to a Data Explorer file (extension `.dx`).

EnSight Case Gold
allows you to specify the scalars you want to write, the location from which the values of scalar functions are to be taken, and the file format, to be exported to an EnSight file (extension `.geo`, `.vel`, `.scl1`, or `.encas`).

FAST
allows you to specify the scalars you want to write, to be exported as a grid file (Plot3D format), a velocity file, and a scalar file. This option is available only for a triangular or tetrahedral mesh.

FAST Solution
allows you to export a single file containing density, velocity, and total energy data. This option is available only for a triangular or tetrahedral mesh.

Fieldview Unstructured
allows you to specify the scalars you want to write, to be exported to a

I-deas Universal
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to an I-deas Universal file.

Mechanical APDL Input
allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to a Mechanical APDL Input file (extension `.cdb`).

NASTRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a NASTRAN file (extension `.bdf`).

PATRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a PATRAN neutral file (extension `.out`).

RadTherm
allows you to specify the surface(s) for which you want to write data and the method of writing the heat transfer coefficient, to be exported to a PATRAN neutral file (extension `.neu`). This option is available only when the **Energy Equation** is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tecplot
allows you to specify the surface(s) and the scalars you want to write, to be exported to a Tecplot file.

Surfaces
specifies the surfaces for which data is to be written for an **ABAQUS**, **ASCII**, Data Explorer, I-deas Universal, **Mechanical APDL Input**, NASTRAN, PATRAN, **RadTherm**, or Tecplot file.

Quantities
specifies valid quantities for output. The attributes of the list are modified based on the active file type. The list may be a single-selection or a multiple-selection list or it may be disabled, depending on the selected **File Type**.

Analysis (for ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN formats)
specifies the finite element analysis intended.

Structural
specifies structural analysis and allows you to select the **Structural Loads** to be written.

Thermal
specifies thermal analysis and allows you to select the **Thermal Loads** to be written.

Structural Loads
contains optional structural loads that can be written to **ABAQUS**, I-deas Universal, **Mechanical APDL Input**, NASTRAN, and PATRAN files. This option is available only when **Structural** analysis is selected.

Force
enables force to be written as a load for a structural analysis.

Pressure
enables pressure to be written as a load for a structural analysis.

Temperature
enables temperature to be written as a load for a structural analysis. This option is available only when the **Energy Equation** is enabled.

Thermal Loads
contains optional thermal loads that can be written to **ABAQUS**, I-deas Universal, **Mechanical APDL Input**, NASTRAN, and PATRAN files. This option is available only when **Thermal** analysis is selected.

Temperature
enables temperature to be written as a load for a thermal analysis.

Heat Flux
enables heat flux to be written as a load for a thermal analysis.

Heat Trans Coeff
enables heat transfer coefficient to be written as a load for a thermal analysis.

Location (for ASCII, CGNS, and EnSight Case Gold formats)
specifies the location from which the values of scalar functions are to be taken.

Node
specifies that data values at the node points are to be exported.

Cell Center
specifies that data values from the cell centers are to be exported.

Format (for CFD-Post Compatible and EnSight Case Gold)
specifies the file format.

Binary
specifies the file format as binary.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASCII
specifies the file format as ASCII.

Heat Transfer Coefficient (for RadTherm format only)
specifies the basis for the heat transfer coefficient exported.

Flux Based
specifies the flux based method for writing the heat transfer coefficient.

Wall Function
specifies the wall function based method for writing the heat transfer coefficient.

Frequency (Time Steps)
specifies the frequency for appending the data during the solution process.

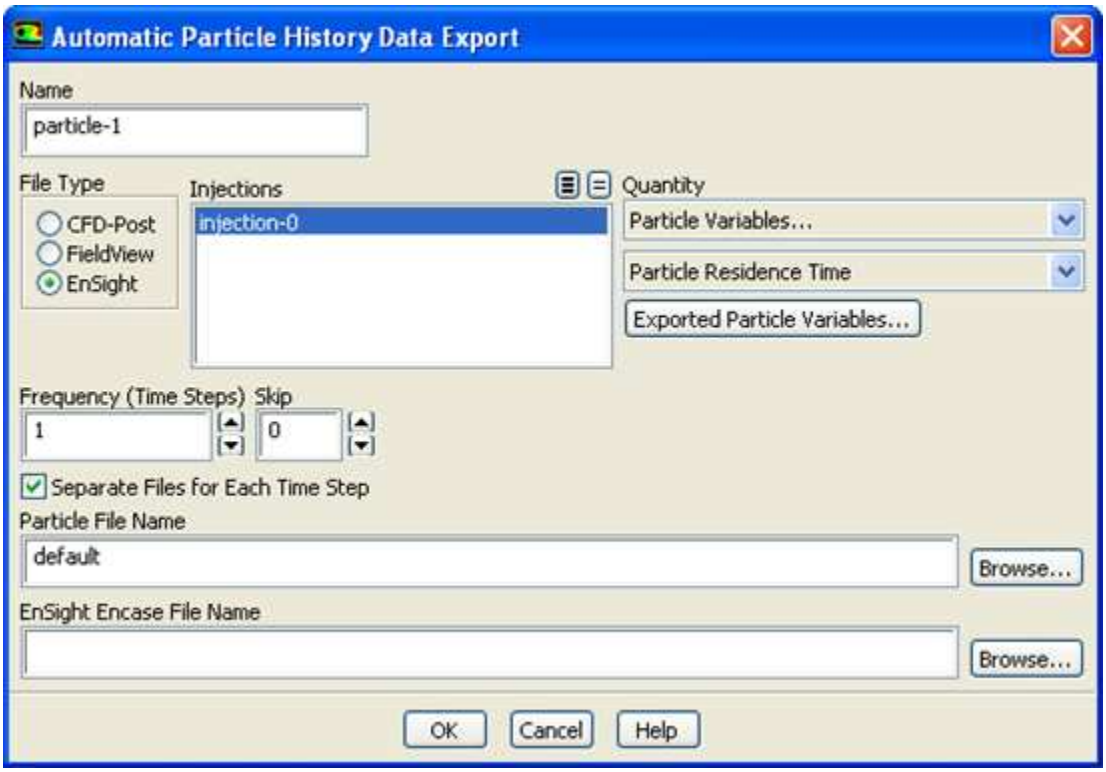
File Name
specifies the root name for the files to be saved.

Append File Name with
allows you to select **flow-time** or **time-step** to be appended to the file name.

Decimal Places in File Name
allows you to specify the number of decimal digits in the file name. This option is available only when **flow-time** is selected in the **Append File Name with** drop-down list. The default value for this field is set to 6.

35.16.4. Automatic Particle History Data Export Dialog Box

The **Automatic Particle History Data Export** dialog box allows you to create an automatic particle history export definition for solution data. See *Creating Automatic Export Definitions for Transient Particle History Data* (p. 102) for details.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
specifies the name of the particle history export definition.

File Type
specifies the type of the file you want to write.

CFD-Post
allows you to write the file in CFD-Post particle tracks format, which can be read in **CFD-Post**.

FieldView
allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

EnSight
allows you to write the file in **EnSight** format.

Injections
allows you to select the required injection from the list of predefined injections.

Quantity
contains the list of variables for which you can export the particle data.

Skip
allows you to "thin" or "sample" the number of particles that are exported.

Frequency (Time Steps) or (DPM Iterations)
specifies the frequency of particle time steps that are used for saving the export file.

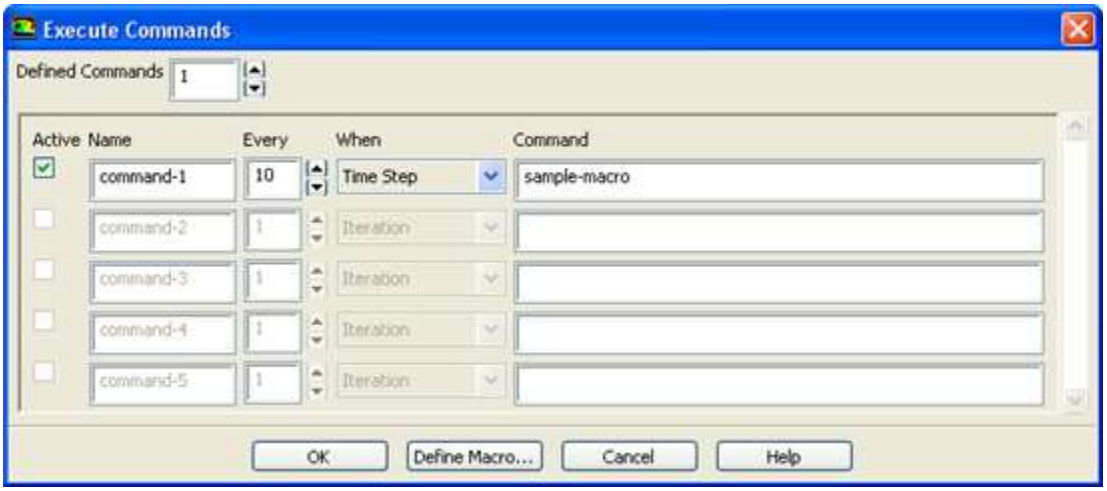
Separate Files for Each Time Step
allows you to have separate exported data files for each time step. Available only when **EnSight** is selected as the **File Type**.

Particle File Name
allows you to specify the file name/directory for the exported data, using the **Browse...** button.

Ensight Encas File Name
is the file name you will specify if you selected **EnSight** under **File Type**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** menu option.

35.16.5. Execute Commands Dialog Box

The **Execute Commands** dialog box allows you to define commands to be executed during the calculation. See *Executing Commands During the Calculation* (p. 1288) for details about using this feature.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defined Commands
sets the total number of monitor commands to be defined.

Active
activates/deactivates the execution of each command.

Name
specifies a name for each command.

Every, When
indicate how often the command is to be executed. You can enter the interval under **Every**and select **Iteration** or **Time Step** under **When**. (**Time Step** is a valid choice only if you are calculating unsteady flow.)

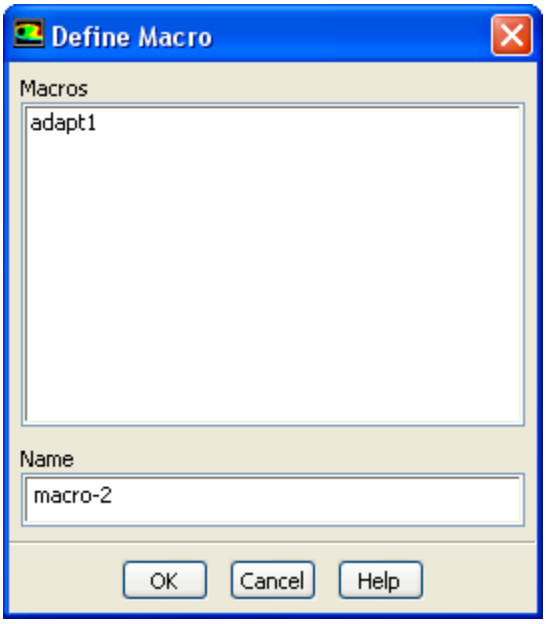
Command
specifies the command to be executed. You can enter text commands or the name of a command macro that you have defined in the *Define Macro Dialog Box* (p. 1951).

Define Macro...
opens the *Define Macro Dialog Box* (p. 1951), in which you can define command macros.

End Macro
ends the definition of a macro. (This button will replace the **Define Macro...**button when you click **OK** in the **Define Macro** dialog box.)

35.16.6. Define Macro Dialog Box

The **Define Macro** dialog box allows you to define macros for automatic execution by the command monitor, or for interactive use by you. See *Defining Macros* (p. 1290) for details.



Controls

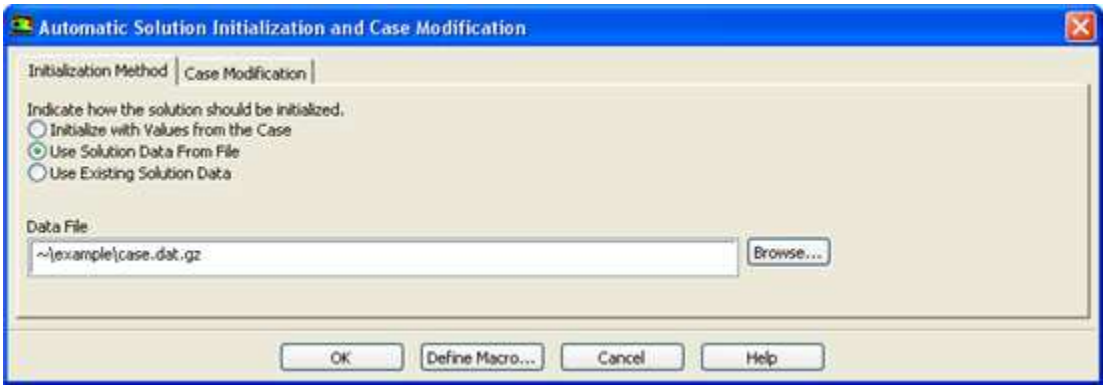
Macros
contains a selectable list of the currently-defined macros.

Name
specifies a name for the command macro.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.16.7. Automatic Solution Initialization and Case Modification Dialog Box

The **Automatic Solution Initialization and Case Modification** dialog box allows you to specify the solution initialization method and to modify the case. See *Automatic Initialization of the Solution and Case Modification* (p. 1292) for details.



Controls

Initialization Method
tab contains several choices for initializing the solution.

Initialize with Values from the Case
uses the values set in the **Solution Initialization** task page.

Use Solution Data From File
requires you to read in a data file containing the desired initialization for the case.

Use Solution Data From Previous Parametric Run with Workbench
(transient cases only) requires you to select one of two methods to initialize the first run. You can **Initialize with Values from the Case**, which uses the values set in the **Solution Initialization** task page. Otherwise, you can **Use Solution Data from File**, which requires you to read in a data file containing the desired initialization for the case.

Use Existing Solution Data
is analogous to changing the values in a case and continuing the calculation. However, the iteration counter will be reset to 0 so that the modifications can be applied. Use this method when no solution data exists, similar to the first run.

Case Modification
allows you to indicate how long you would like to run with the original settings, then make any modifications to the case settings.

Defined Modifications
indicates the number of modifications for the case file.

Active
allows you to enable or disable a defined case modification.

Name
represents the name of the case modification.

Commands
represents an area where you can input text commands.

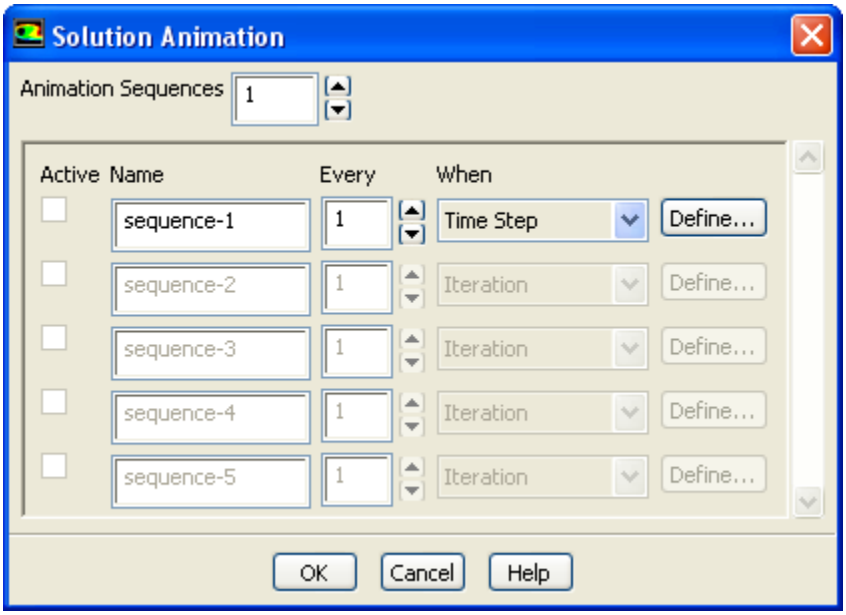
Number of Iterations/Time Steps
represents the number of iterations or time steps that you want to run define ed case modification commands.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define Macro...
opens the *Define Macro Dialog Box* (p. 1951), in which you can define command macros.

35.16.8. Solution Animation Dialog Box

You can use the **Solution Animation** dialog box to create an animation sequence and indicate how often each frame of the sequence should be created. See *Defining an Animation Sequence* (p. 1296) for details about the items below.



Controls

Animation Sequences
sets the total number of animation sequences to be defined.

Active
activates/deactivates each animation sequence.

Name
specifies a name for each animation sequence.

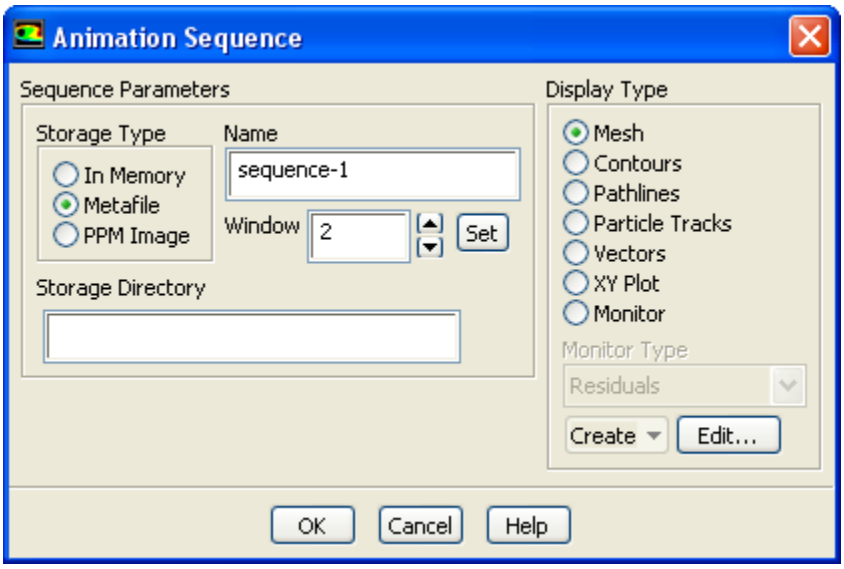
Every, When
indicate how often you want to create a new frame in the animation sequence. You can enter the interval under **Every** and select **Iteration** or **Time Step** in the drop-down list below **When**. (**Time Step** is a valid choice only if you are calculating unsteady flow.)

Define...
opens the *Animation Sequence Dialog Box* (p. 1953), in which you can define an animation sequence.

35.16.9. Animation Sequence Dialog Box

The **Animation Sequence** dialog box allows you to define each animation sequence. See *Defining an Animation Sequence* (p. 1296) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Sequence Parameters

contains general parameters for the storage and display location of the animation sequence.

Storage Type

specifies whether you want ANSYS FLUENT to save the animation sequence frames in memory (**In Memory**) or on your computer's hard drive (**Metafile** or **PPM Image**).

Name

specifies the name of the sequence.

Window

specifies the ID of the graphics window where you want the plot to be displayed. You must click **Set** to set the specified **Window**.

When ANSYS FLUENT is iterating, the active graphics window is set to this window to update the plot. If you want to maintain each animation in a separate window, specify a different **Window** ID for each.

Set

sets the specified **Window** to be the window where the plot will be displayed. (The specified window will open, if it is not already open.)

Storage Directory

specifies the directory where you want to store the files. (This can be a relative or absolute path.)

Display Type

specifies the type of display to be animated.

Mesh

opens the *Mesh Display Dialog Box* (p. 1635) where you can select a mesh display.

Contours

opens the *Contours Dialog Box* (p. 1967) where you can select a contour display.

Pathlines

opens the *Pathlines Dialog Box* (p. 1975) where you can select pathlines to display.

Particle Tracks

opens the *Particle Tracks Dialog Box* (p. 1980) where you can select particle tracks to display.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vectors
opens the *Vectors Dialog Box* (p. 1970) where you can select a vector display.

XY Plot
opens the *Solution XY Plot Dialog Box* (p. 2012) where you can select a solution XY plot.

Monitor
selects a monitor plot.

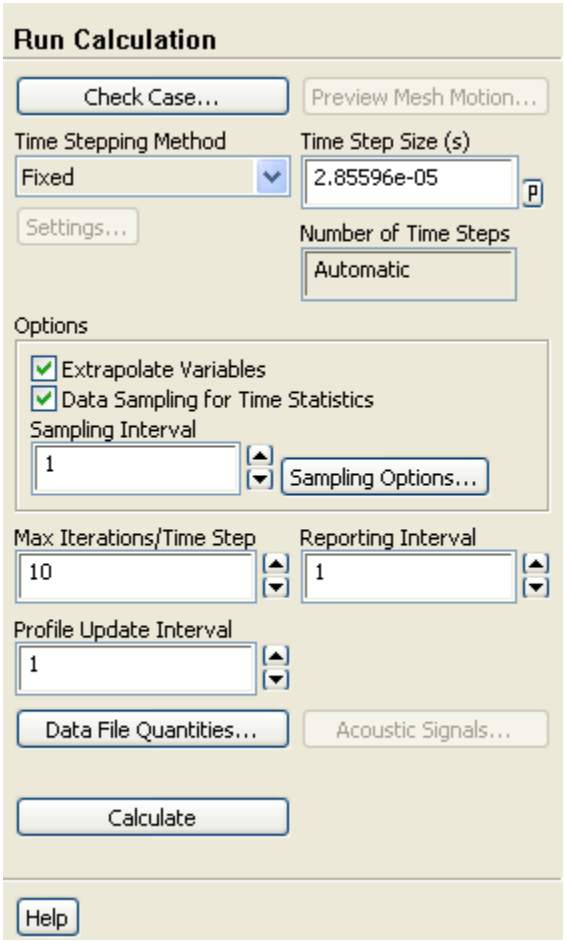
Monitor Type
contains a drop-down list of the available types of monitor plots (**Residuals**, **Drag**, **Lift**, **Moment**,and **Statistics**).

Create
provides a drop-down list that contains options for creating surface monitors or volume monitors.

Edit...
opens the dialog box that corresponds to the selected display or monitor type (e.g., the **Contours** dialog box if you have selected **Contours**).

35.17. Run Calculation Task Page

The **Run Calculation** task page allows you to start the solver iterations. See *Performing Steady-State Calculations* (p. 1247) and *Performing Time-Dependent Calculations* (p. 1255) for details about the items below.



Controls

Check Case...
opens the *Case Check Dialog Box* (p. 1959).

Preview Mesh Motion...
opens the *Mesh Motion Dialog Box* (p. 1896).

Pseudo Transient Options
is available for fluid and solid zones and will only appear for steady-state cases, where the pressure-based coupled solver is used. Note that the **Pseudo Transient** option must be enabled in the *Solution Methods Task Page* (p. 1900). For details about the available options, see *Performing Pseudo Transient Calculations* (p. 1249).

Number of Iterations
(for steady flow calculations) sets the number of iterations to be performed. (For unsteady calculations using the explicit unsteady formulation, this will specify the number of time steps, since each iteration will be a time step.)

Time Stepping Method
(for transient flow calculations) contains options for how the time step is determined.

Fixed
selects a fixed time step, equal to the specified **Time Step Size**.

Adaptive
selects a time step that is initially equal to the specified **Time Step Size**, but gets modified by ANSYS FLUENT as the calculation proceeds. See *Adaptive Time Stepping* (p. 1264) for details.

Variable
enables variable time stepping method. The inputs are same as for the adaptive time stepping method, with the exception of specifying a global Courant number. **Variable** is selectable when the VOF multiphase model is enabled. See *Variable Time Stepping* (p. 1266) for details.

Settings...
opens either the *Adaptive Time Step Settings Dialog Box* (p. 1960) when **Adaptive** is selected as the **Time Stepping Method**, or the *Variable Time Step Settings Dialog Box* (p. 1962) when **Variable** is selected as the **Time Stepping Method**.

Time Step Size(s)
(for transient flow calculations) sets the magnitude of the (physical) time step Δt.

This quantity can be assigned to an input parameter (indicated by the small "p" icon next to the field). Clicking this icon displays the *Select Input Parameter Dialog Box* (p. 1807) where you can create and assign input parameters.

Time Step Size for Acoustic Data Export
determines the highest frequency that the acoustic analysis reproduces. This is available when the FW-H acoustics model is enabled.

Number of Time Steps
(for transient flow calculations) sets the number of time steps to be performed.

Options
contains options related to unsteady calculations.

Extrapolate Variables
instructs ANSYS FLUENT to predict the solution variable values for the next time step and then input that predicted value as an initial guess for the inner iterations of the current time step.

Data Sampling for Time Statistics
enables the sampling of data during an unsteady calculation. See *User Inputs for Time-Dependent Problems* (p. 1256) and *Postprocessing for Time-Dependent Problems* (p. 1267) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sampling Interval
allows you to specify the frequency of the **Data Sampling for Time Statistics**.

Sampling Options...
opens the *Sampling Options Dialog Box* (p. 1963) for the **Data Sampling forTime Statistics**.

Max Iterations/Time Step
for unsteady flow calculations using an implicit unsteady formulation) sets the maximum number of iterations to be performed per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step. See *User Inputs for Time-Dependent Problems* (p. 1256) for details.

Reporting Interval
sets the number of iterations that will pass before convergence monitors will be printed and plotted. The default is 1 (i.e., reports will be updated after each iteration).

Postprocess Pollutants
results in the automatic postprocessing of pollutants during a transient simulation. This option is available when one of the pollution models is enabled.

Max Post Iterations/Time Step
sets the maximum number of postprocessing iterations to be performed per time step. This field is available when the **Postprocess Pollutants** option is enabled.

Profile Update Interval
sets the number of iterations that will pass before user-defined functions for boundary profiles, properties, etc. will be updated.

Solution Steering
allows you to set parameters that will help ANSYS FLUENT guide the calculations to a converged solution (available only for steady-state flows in the density-based implicit solver). See *Solution Steering* (p. 1320) for more information. When enabled, the following options are available:

Flow Type
allows you to select the flow type that best describes the flow in the solution domain. Five choices are available: **incompressible**, **subsonic**, **transonic**, **supersonic**, and **hypersonic**.

Use FMG Initialization
allows for full multigrid initialization.

First to Higher Order Blending
allows you to reduce the desired solution accuracy by selecting a blending factor less than 100%. The default setting is 100%. See First-to-Higher Order Blending in the Theory Guide for more information. The blending factor will be grayed out if **Second Order Upwind** discretization for the **Flow** equations is not selected in the **Solution Methods** task page. The solution accuracy may be reduced (typical values are 75% or 50%) if it is not possible to obtain a converged solution with the maximum second-order accuracy (i.e. blending = 100%)

More Settings...
opens the *Solution Steering Dialog Box* (p. 1958).

Courant Number
is a non-adjustable field displaying the current CFL number, which allows you to view it during the calculation.

Data File Quantities...
opens the *Data File Quantities Dialog Box* (p. 1945).

Acoustic Signals...
opens the *Acoustic Signals Dialog Box* (p. 1963). This button is only available when the acoustics model is enabled.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculate

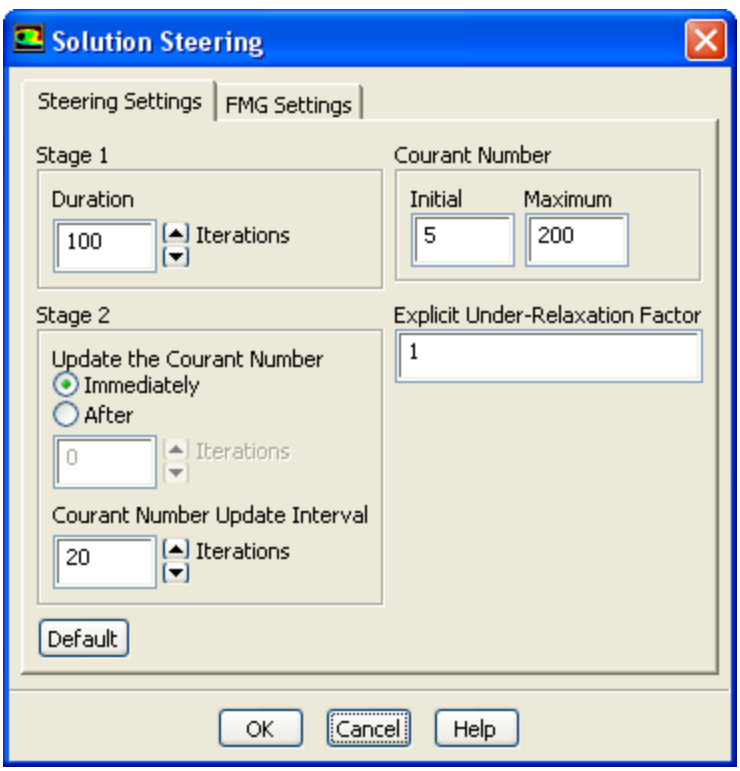
starts the calculations. While the calculation is in progress, a **Working** dialog box will appear. Clicking the **Cancel** button or typing `<Control-C>` in the console window will interrupt the calculation (as soon as it is safe to stop).

For additional information, please see the following sections:

35.17.1. Solution Steering Dialog Box
35.17.2. Case Check Dialog Box
35.17.3. Adaptive Time Step Settings Dialog Box
35.17.4. Variable Time Step Settings Dialog Box
35.17.5. Sampling Options Dialog Box
35.17.6. Acoustic Signals Dialog Box

35.17.1. Solution Steering Dialog Box

The **Solution Steering** dialog box is used to set the parameters that control the solution steering strategy. Solution steering will typically perform full multigrid (FMG) initialization followed by two iterative stages (Stage 1 and Stage 2). The purpose of Stage 1 is to navigate the solution from the difficult initial phase of the solution toward convergence by insuring maximum stability. During this stage, the solution is advanced gradually from 1st-order accuracy to maximum accuracy (user specified and typically 2nd-order) at a constant low CFL value. In Stage 2, the solution is driven hard towards convergence by regular adjustments of the CFL value to insure fast convergence as well as to prevent possible divergence. In Stage 2, the residual history is monitored and analyzed through regular intervals to determine if an increase or decrease in CFL value is needed to obtain fast convergence or to prevent divergence. See *Solution Steering* (p. 1320) for details.



Controls

Steering Settings

allows you to modify the steering parameters used in Stages 1 and 2.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage 1
allows you to set parameters relating to stage 1.

Duration
is the number of iterations in stage 1. The CFL number used during these iterations is set in the **Initial** field, in the **Courant Number** group box.

Stage 2
allows you to set parameters relating to stage 2.

Update the Courant Number
allows you to update the Courant number either **Immediately**, or **After** a specified number of iterations.

Courant Number Update Interval
defines the frequency at which the Courant number is updated.

Courant Number
allows you to set the starting (**Initial**) and maximum allowed (**Maximum**) Courant number values. The solution steering algorithm will not allow the solver to exceed the maximum Courant number, but will allow the solver to use a Courant number less than the initial Courant number if divergence in the solution has occurred.

Explicit Under-Relaxation Factor
allows the solution to be under-relaxed to improve convergence.

Default
resets any changes made to the parameters to their original default values.

FMG Settings
allows you to set FMG parameters. For more information about FMG initialization, refer to *Full Multigrid (FMG) Initialization* (p. 1244).

Number of Multigrid Levels
allows you to set the number of multigrid levels.

Number of Cycles
allows you to set the number of cycles for a selected level.

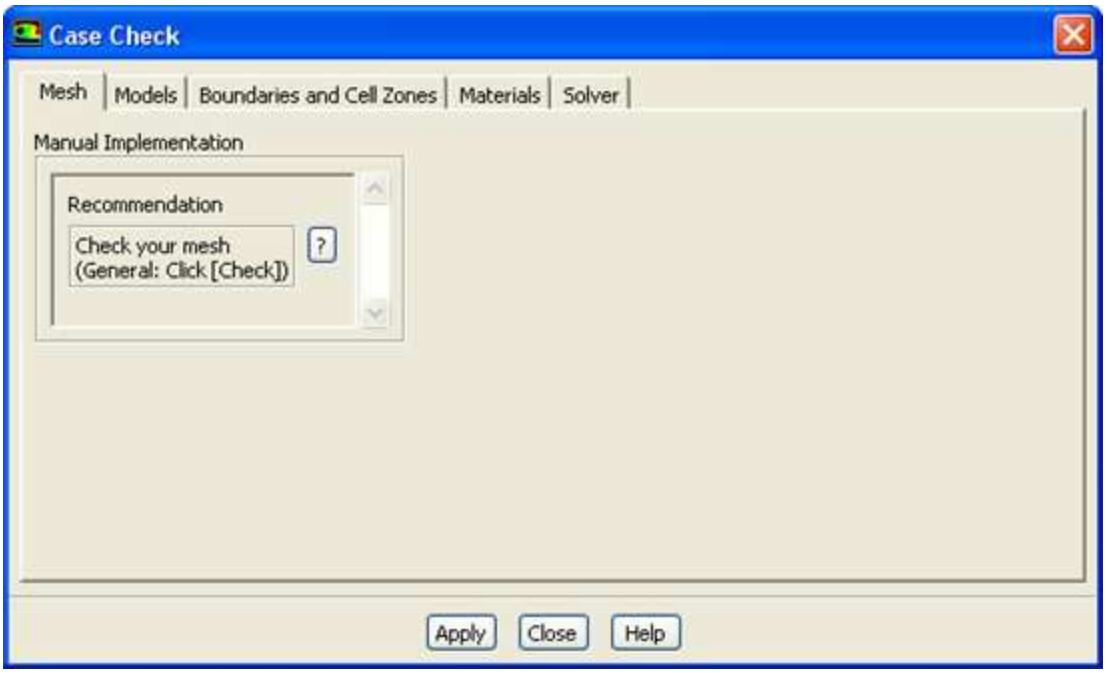
FMG Courant Number
allows you to set the FMG Courant number.

Default
resets any changes to the original default values.

35.17.2. Case Check Dialog Box

This function provides you with guidance and best practices when choosing case parameters and models. Your case will be checked for compliance in the mesh, models, boundary and cell zone conditions, material properties, and solver categories. Established rules are available for each category, with recommended changes to your current settings. Information about each of the recommendations is available in *Checking Your Case Setup* (p. 1304).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mesh
displays recommendations, if any, relating the to the mesh used in the case. See *Checking the Mesh* (p. 1306) for details.

Models
displays recommendations, if any, relating the to the models used in the case. See *Checking Model Selections* (p. 1308) for details.

Boundaries and Cell Zones
displays recommendations, if any, relating the to the cell zones or boundaries defined in the case. See *Checking Boundary and Cell Zone Conditions* (p. 1310) for details.

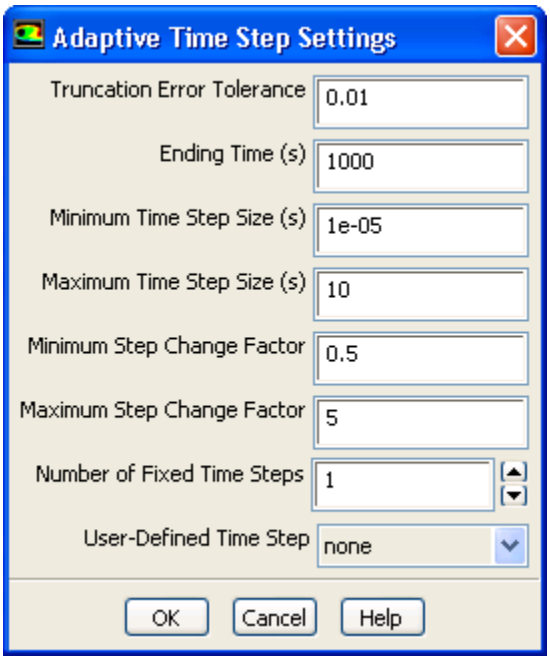
Materials
displays recommendations, if any, relating the to the materials defined in the case. See *Checking Material Properties* (p. 1313) for details.

Solver
displays recommendations, if any, relating the to the solver settings used in the case. See *Checking the Solver Settings* (p. 1314) for details.

35.17.3. Adaptive Time Step Settings Dialog Box

The **Adaptive Time Step Settings** dialog box is used to set the parameters that control the adaptive time stepping. See *Adaptive Time Stepping* (p. 1264) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Truncation Error Tolerance

specifies the threshold value to which the computed truncation error is compared. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

Ending Time

specifies an ending time for the calculation. Since the ending time cannot be determined by multiplying the number of time steps by a fixed time step size, you need to specify it explicitly.

Minimum/Maximum Time Step Size

specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Minimum/Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1 < f < f_{max}$, Δt_n is increased to meet the required tolerance.
- If $1 < f_{max} < f$, Δt_n is increased, but its maximum possible value is $f_{max} \Delta t_{n-1}$.
- If $f_{min} < f < 1$, Δt_n is unchanged.
- If $f < f_{min} < 1$, Δt_n is decreased.

Number of Fixed Time Steps

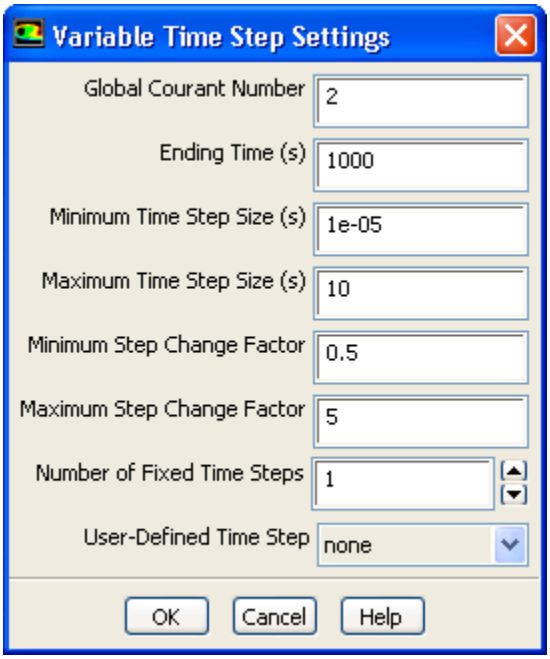
specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Time Step Size** under **Time**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Time Step
contains a drop-down list of available user-defined functions.

35.17.4. Variable Time Step Settings Dialog Box

The **Variable Time Step Settings** dialog box contains parameters related to variable time stepping. See *Variable Time Stepping* (p. 1266) for details.



Controls

Global Courant Number
allows you to specify the Courant number. The default value for the **Global Courant number** is 2.

Ending Time
specifies an ending time for the calculation. Since the ending time cannot be determined by multiplying the number of time steps by a fixed time step size, you need to specify it explicitly.

Minimum/Maximum Time Step Size
specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Minimum/Maximum Step Change Factor
limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1 < f < f_{max}$, Δt_n is increased to meet the required tolerance.
- If $1 < f_{max} < f$, Δt_n is increased, but its maximum possible value is $f_{max} \Delta t_{n-1}$.
- If $f_{min} < f < 1$, Δt_n is unchanged.
- If $f < f_{min} < 1$, Δt_n is decreased.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Fixed Time Steps

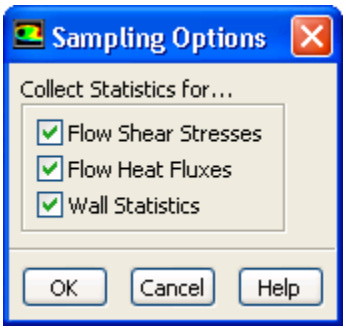
specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Time Step Size** under **Time**.

User-Defined Time Step

contains a drop-down list of available user-defined functions.

35.17.5. Sampling Options Dialog Box

The **Sampling Options** dialog box allows you to specify a collection of statistics for: **Flow Shear Stresses**, **Flow Heat Fluxes**, or **Wall Statistics**.



Controls

Flow Shear Stresses

allows you to enable or disable the flow shear stress statistics used in postprocessing.

Flow Heat Fluxes

allows you to enable or disable the flow heat fluxes statistics used in postprocessing.

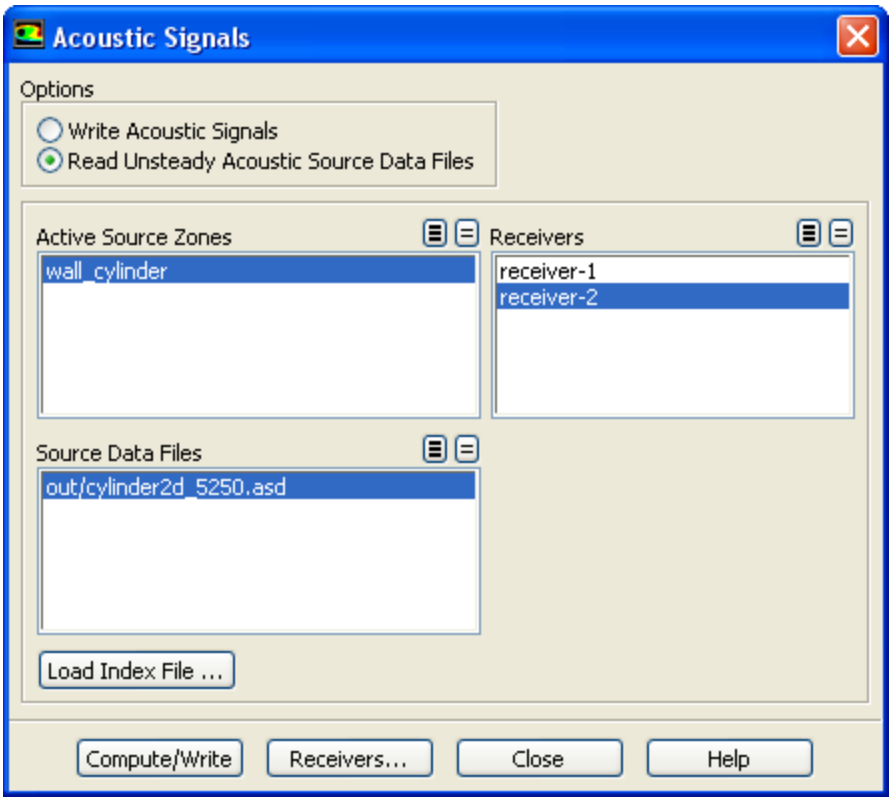
Wall Statistics

allows you to enable or disable the wall statistics used in postprocessing.

35.17.6. Acoustic Signals Dialog Box

The **Acoustic Signals** dialog box is used to compute and save the sound pressure signals. See *Postprocessing the FW-H Acoustics Model Data* (p. 1000) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the options available for acoustic signal postprocessing.

Write Acoustic Signals

enables the parameters needed to write the sound pressure data to files.

Read Unsteady Acoustic Source Data Files

enables the parameters needed to compute the sound pressure signals using the source data saved to files.

Active Source Zones

contains source zones you want to include to compute sound. See *Specifying Source Surfaces* (p. 995) for details.

Receivers

contains all the receivers for which you can compute sound.

Source Data Files

contains all the source data files that you can use to compute sound.

Load Index File...

opens the **Select File** dialog box, in which you can select the index file for your computation.

Write

writes the sound pressure data. This button will appear only when **Write Acoustic Signals** is selected under **Options**.

Compute/Write

computes and saves the sound pressure data. This button will appear only when **Read Unsteady Acoustic Source Data Files** is selected under **Options**.

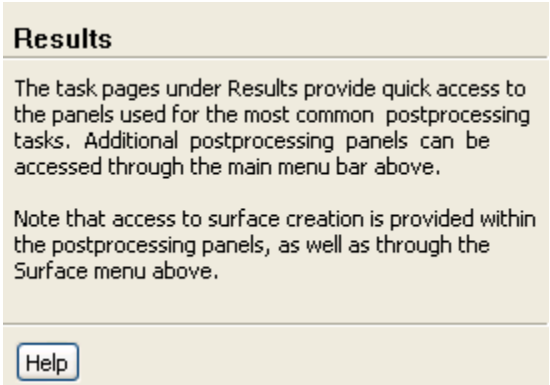
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Receivers...

opens the *Acoustic Receivers Dialog Box* (p. 1734) in which you can define additional receivers.

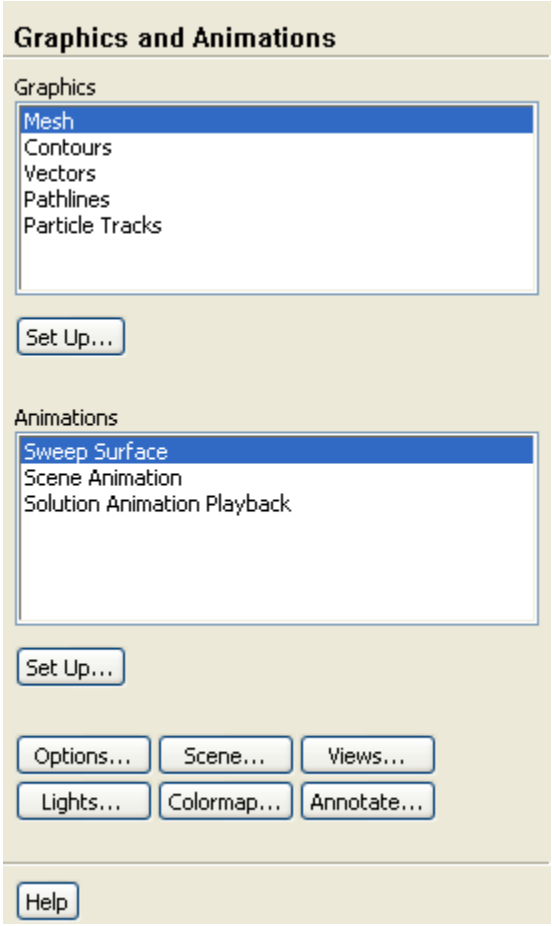
35.18. Results Task Page

The **Results** task page introduces you to the main tasks involved in setting up and displaying the results of your CFD simulation using ANSYS FLUENT.



35.19. Graphics and Animations Task Page

The **Graphics and Animation** task page allows you to visualize the results of your CFD simulation by allowing you to set up plots of contours, vectors, pathlines, particle tracks, scene descriptions and animations. See *Displaying Graphics* (p. 1379) for more information.



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Graphics
displays a list of the available graphics objects.

You can double-click an item in the **Graphics** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Mesh
- selecting this item and clicking the **Set Up...** button opens the *Mesh Display Dialog Box* (p. 1635).

Contours
- selecting this item and clicking the **Set Up...** button opens the *Contours Dialog Box* (p. 1967).

Vectors
- selecting this item and clicking the **Set Up...** button opens the *Vectors Dialog Box* (p. 1970).

Pathlines
- selecting this item and clicking the **Set Up...** button opens the *Pathlines Dialog Box* (p. 1975).

Particle Tracks
- selecting this item and clicking the **Set Up...** button opens the *Particle Tracks Dialog Box* (p. 1980).

Set Up...
opens the dialog box corresponding to the selected object in the **Graphics** list.

Animations
displays a list of the available animation objects.

You can double-click an item in the **Animations** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Sweep Surface
- selecting this item and clicking the **Set Up...** button opens the *Sweep Surface Dialog Box* (p. 1984).

Scene Animation
- selecting this item and clicking the **Set Up...** button opens the *Animate Dialog Box* (p. 1986).

Solution Animation
- selecting this item and clicking the **Set Up...** button opens the *Playback Dialog Box* (p. 1990).

Set Up...
opens the dialog box corresponding to the selected object in the **Animations** list.

Options...
opens the *Display Options Dialog Box* (p. 1992).

Scene...
opens the *Scene Description Dialog Box* (p. 1994).

Views...
opens the *Views Dialog Box* (p. 2000).

Lights...
opens the *Lights Dialog Box* (p. 2005).

Colormap...
opens the *Colormap Dialog Box* (p. 2007).

Annotate...
opens the *Annotate Dialog Box* (p. 2010).

For additional information, please see the following sections:

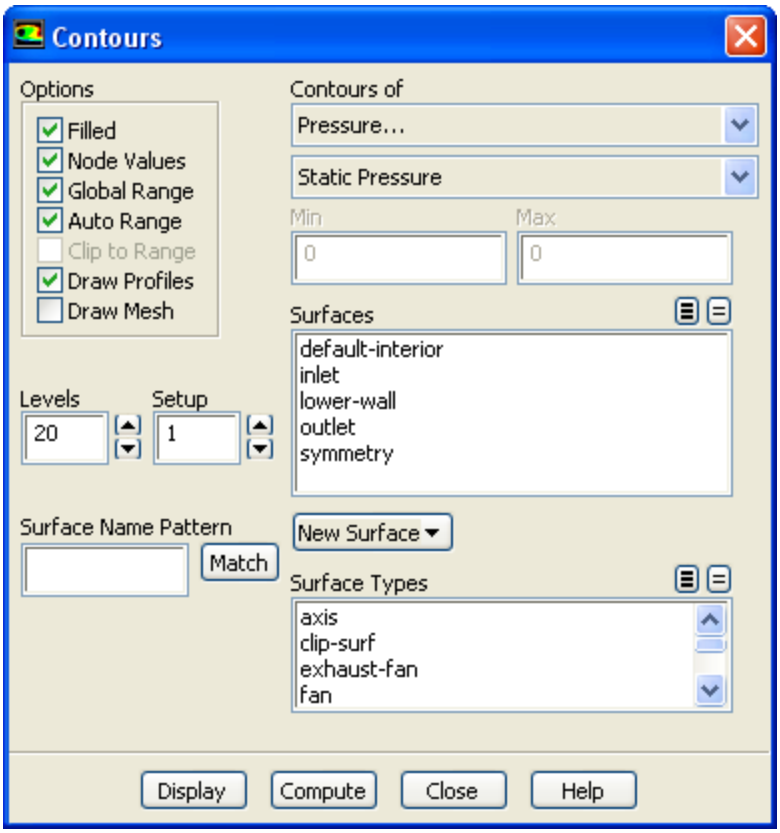
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.19.1. Contours Dialog Box
35.19.2. Profile Options Dialog Box
35.19.3. Vectors Dialog Box
35.19.4. Vector Options Dialog Box
35.19.5. Custom Vectors Dialog Box
35.19.6. Vector Definitions Dialog Box
35.19.7. Pathlines Dialog Box
35.19.8. Path Style Attributes Dialog Box
35.19.9. Ribbon Attributes Dialog Box
35.19.10. Particle Tracks Dialog Box
35.19.11. Reporting Variables Dialog Box
35.19.12. Sweep Surface Dialog Box
35.19.13. Create Surface Dialog Box
35.19.14. Animate Dialog Box
35.19.15. Save Picture Dialog Box
35.19.16. Playback Dialog Box
35.19.17. Display Options Dialog Box
35.19.18. Scene Description Dialog Box
35.19.19. Display Properties Dialog Box
35.19.20. Transformations Dialog Box
35.19.21. Iso-Value Dialog Box
35.19.22. Pathline Attributes Dialog Box
35.19.23. Bounding Frame Dialog Box
35.19.24. Views Dialog Box
35.19.25. Write Views Dialog Box
35.19.26. Mirror Planes Dialog Box
35.19.27. Graphics Periodicity Dialog Box
35.19.28. Camera Parameters Dialog Box
35.19.29. Lights Dialog Box
35.19.30. Colormap Dialog Box
35.19.31. Colormap Editor Dialog Box
35.19.32. Annotate Dialog Box

35.19.1. Contours Dialog Box

The **Contours** dialog box controls the display of contour and profile plots. See *Displaying Contours and Profiles* (p. 1386) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the check buttons that set various contour display options.

Filled

toggles between filled contours and line contours.

Node Values

toggles between using scalar field values at nodes and at cell centers for computing the contours. When the **Filled** option is off, **Node Values** is always on. See *Choosing Node or Cell Values* (p. 1392) for details.

Global Range

toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range

toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range

determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See *Specifying the Range of Magnitudes Displayed* (p. 1390) for details.

Draw Profiles

causes the addition of a profile plot to the contour plot. The *Profile Options Dialog Box* (p. 1970) is opened when **Draw Profiles** is selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Draw Mesh
toggles between displaying and not displaying the mesh. The *Mesh Display Dialog Box* (p. 1635) is opened when **Draw Mesh** is selected.

Levels
sets the number of contour levels that are displayed.

Setup
indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the **Contours** dialog box. See *Storing Contour Plot Settings* (p. 1393) for details.

Surface Name Pattern
specifies the pattern to look for in the names of surfaces. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Surfaces** list with names that match the specified pattern. See *Generating Contour and Profile Plots* (p. 1387) for information about matching additional characters using * and ?.

Contours of
contains a list from which you can select the scalar field to be contoured.

Min
shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Surfaces
contains a list from which you can select the surfaces on which to draw contours. For 2D cases, if no surface is selected, contouring is done on the entire domain. For 3D cases, you must always select at least one surface.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the *Point Surface Dialog Box* (p. 1927).

Line/Rake
opens the *Line/Rake Surface Dialog Box* (p. 1928).

Plane
opens the *Plane Surface Dialog Box* (p. 1929).

Quadric
opens the *Quadric Surface Dialog Box* (p. 1931).

Iso-Surface
opens the *Iso-Surface Dialog Box* (p. 1933).

Iso-Clip
opens the *Iso-Clip Dialog Box* (p. 1935).

Surface Types
contains a selectable list of types of surfaces. If you select (or deselect) an item in this list, all surfaces of that type will be selected (or deselected) automatically in the **Surfaces** list.

Display
draws the contours in the active graphics window.

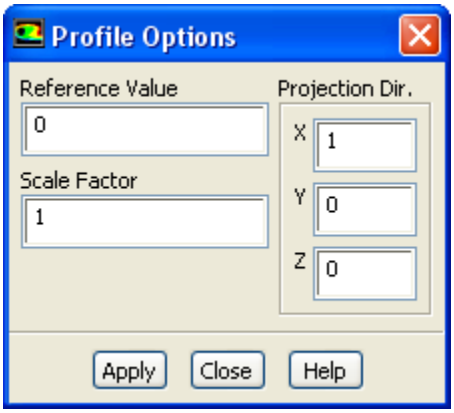
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute

calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

35.19.2. Profile Options Dialog Box

The **Profile Options** dialog box controls the scaling and projection direction of profiles. It is opened from the *Contours Dialog Box* (p. 1967), and you will display the profiles using the **Display** button in that dialog box. See *Displaying Contours and Profiles* (p. 1386) for details about the items below.



Controls

Reference Value

sets the "zero height" reference value for the profile. Any point on the profile with a value equal to the **Reference Value** will be plotted exactly on the defining surface. Values greater than the **Reference Value** will be projected ahead of the surface (in the direction of **Projection Dir.** and scaled by **Scale Factor**), and values less than the **Reference Value** will be projected behind the surface.

Scale Factor

sets the length scale factor for projection. After subtracting off the **Reference Value**, ANSYS FLUENT multiplies the resulting solution value by the **Scale Factor** to form a length.

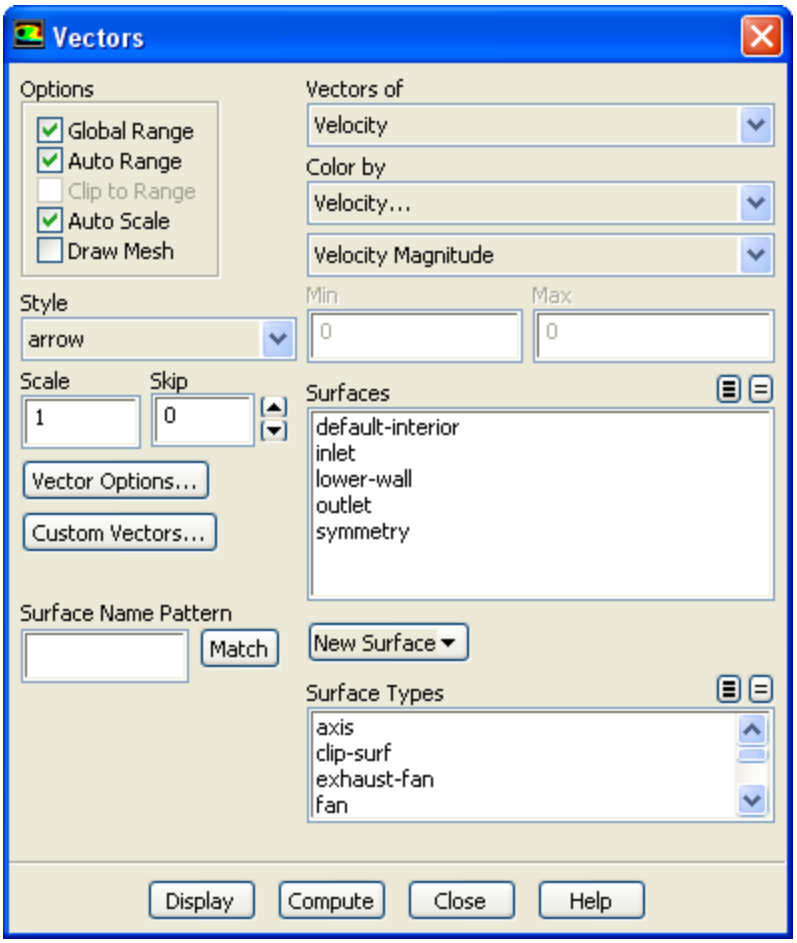
Projection Dir.

sets the direction in which profiles are projected. In 2D, for example, a contour plot of pressure on the entire domain can be projected in the z direction to form a carpet plot, or a contour plot of x velocity on a sequence of x-coordinate slice lines can be projected in the x direction to form a series of velocity profiles.

35.19.3. Vectors Dialog Box

The **Vectors** dialog box controls the display of vector plots. See *Displaying Vectors* (p. 1393) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains check buttons that set various display options.

Global Range

toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range

toggles between automatic and manual setting of the range of scalar field values.

Clip to Range

controls the display of vectors that have a value outside the range specified by **Min** and **Max**. When on, no vectors are displayed outside the range. When off, vectors are displayed outside the range using the colors at the top and bottom of the color scale. This option is applicable only when **Auto Range** is off. See *Specifying the Range of Magnitudes Displayed* (p. 1397) for details.

Auto Scale

enables the scaling of all vectors in the domain such that when the **Scale** is 1, there will be minimal overlap of vectors.

Draw Mesh

toggles between displaying and not displaying the mesh. The *Mesh Display Dialog Box* (p. 1635) is opened when **Draw Mesh** is selected.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Style
selects the style in which the vectors are drawn. Available styles are **cone**, **filled-arrow**, **arrow**, **harpoon**, and **headless**.

Scale
sets the factor by which the vectors should be scaled. See *Scaling the Vectors* (p. 1395) for details.

Skip
allows you to "thin" or "sample" the vectors that are displayed. See *Skipping Vectors* (p. 1396) for details.

Vector Options...
opens the *Vector Options Dialog Box* (p. 1973), in which you can set additional options for vector displays.

Custom Vectors...
opens the *Custom Vectors Dialog Box* (p. 1973), in which you can define your own vector fields.

Surface Name Pattern
specifies the pattern to look for in the names of surfaces. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Surfaces** list with names that match the specified pattern. See *Generating Vector Plots* (p. 1394) for information about matching additional characters using * and ?.

Vectors of
contains a list from which you can select the vector field to be plotted.

Color by
contains a list from which you can select the scalar field by which the vectors are colored.

Min
shows the value to which the lower end of the color scale is mapped. You can set this value manually if **Auto Range** is off.

Max
shows the value to which the upper end of the color scale is mapped. You can set this value manually if **Auto Range** is off.

Surfaces
contains a list from which you can select the surfaces on which to display vectors. In 2D, vectors are displayed on the entire domain if no surface is selected.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the *Point Surface Dialog Box* (p. 1927).

Line/Rake
opens the *Line/Rake Surface Dialog Box* (p. 1928).

Plane
opens the *Plane Surface Dialog Box* (p. 1929).

Quadric
opens the *Quadric Surface Dialog Box* (p. 1931).

Iso-Surface
opens the *Iso-Surface Dialog Box* (p. 1933).

Iso-Clip
opens the *Iso-Clip Dialog Box* (p. 1935).

Surface Types
contains a selectable list of types of surfaces. If you select (or deselect) an item in this list, all surfaces of that type will be selected (or deselected) automatically in the **Surfaces** list.

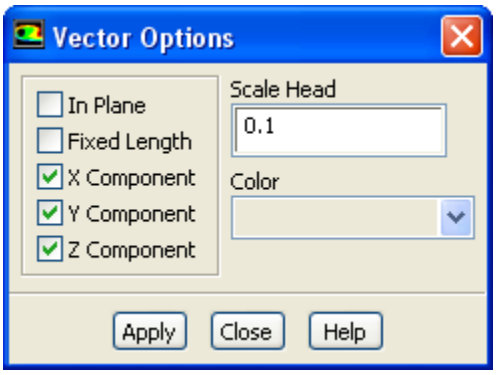
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display
draws the vectors in the active graphics window.

Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

35.19.4. Vector Options Dialog Box

The **Vector Options** dialog box allows you to set additional parameters for vector displays. It is opened from the *Vectors Dialog Box* (p. 1970). See *Vector Plot Options* (p. 1395) for details about the items below.



Controls

In Plane
toggles the display of vector components in the plane of the surface selected for display. This feature is useful for visualizing components that are normal to the flow. See *Drawing Vectors in the Plane of the Surface* (p. 1396) for details.

Fixed Length
enables the display of vectors that are all the same length. See *Displaying Fixed-Length Vectors* (p. 1397) for details.

X, Y, Z Component
toggle the display of the Cartesian components of the vectors. See *Displaying Vector Components* (p. 1397) for details.

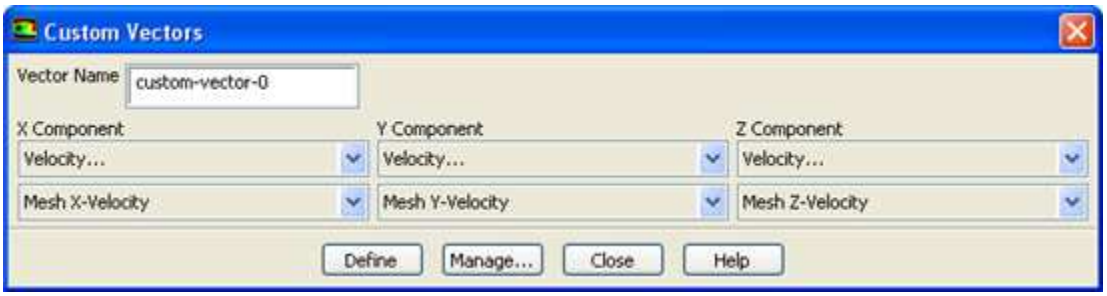
Scale Head
controls the size of the arrowhead on vector styles that include heads.

Color
specifies a single color for the display of all vectors. See *Displaying Vectors Using a Single Color* (p. 1398) for details.

35.19.5. Custom Vectors Dialog Box

The **Custom Vectors** dialog box allows you to define custom vectors based on existing quantities. Any vectors that you define will be added to the **Vectors of** list in the *Vectors Dialog Box* (p. 1970). To open the **Custom Vectors** dialog box, click **Custom Vectors...** in the **Vectors** dialog box. See *Creating and Managing Custom Vectors* (p. 1398) for details about custom vectors.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Vector Name

specifies the name of the vector you are defining. Should you decide to change the name after you have defined the vector, you can do so in the *Vector Definitions Dialog Box* (p. 1974), which you can open by clicking on the **Manage...** button.

X, Y, Z Component

specify the x, y, and z components of the vector. Each drop-down list contains the available field functions.

Define

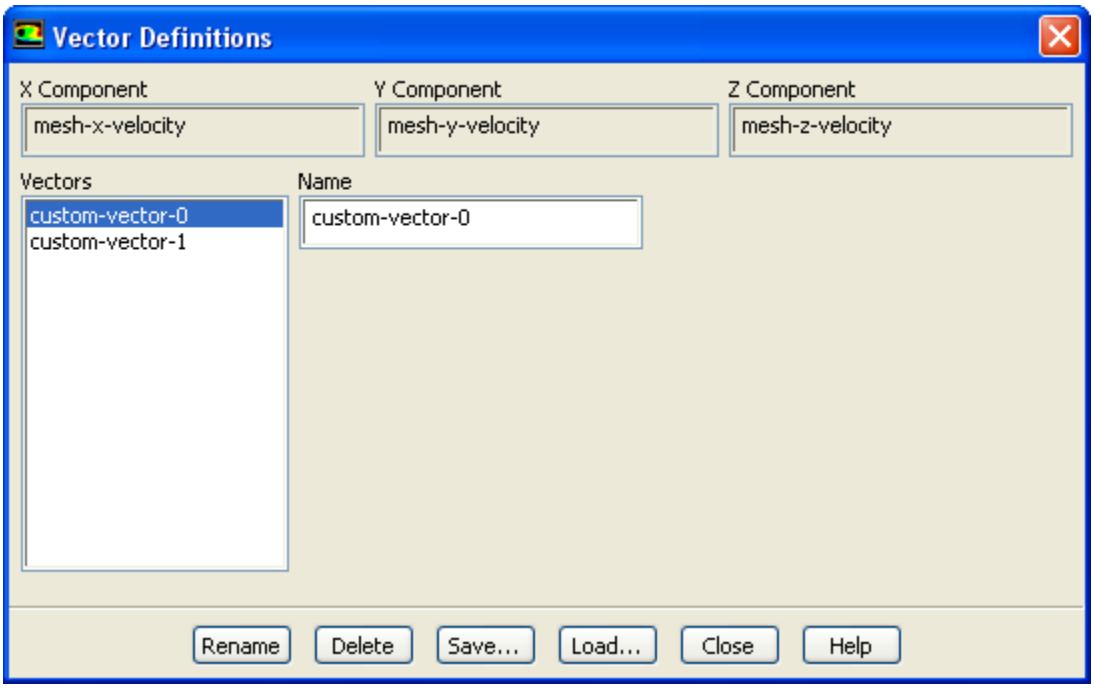
creates the vector and adds it to the **Vectors of** list in the *Vectors Dialog Box* (p. 1970).

Manage...

opens the *Vector Definitions Dialog Box* (p. 1974), which enables you to check, rename, save, load, and delete custom vectors.

35.19.6. Vector Definitions Dialog Box

The **Vector Definitions** dialog box allows you to check, rename, save, load, and delete custom vectors that you defined in the *Custom Vectors Dialog Box* (p. 1973). See *Creating and Managing Custom Vectors* (p. 1398) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X, Y, Z Component

display the x, y, and z components of the vector.

Vectors

contains a selectable list of custom vectors. When you select a vector, its components will appear in the **X**, **Y**, and **Z Component** fields, and its name will appear in the **Name** field.

Name

displays the name of the currently selected vector. You can enter a new name in this box if you want to rename the vector.

Rename

changes the name of the selected function to the name specified in the **Name** field.

Delete

deletes the selected vector.

Save...

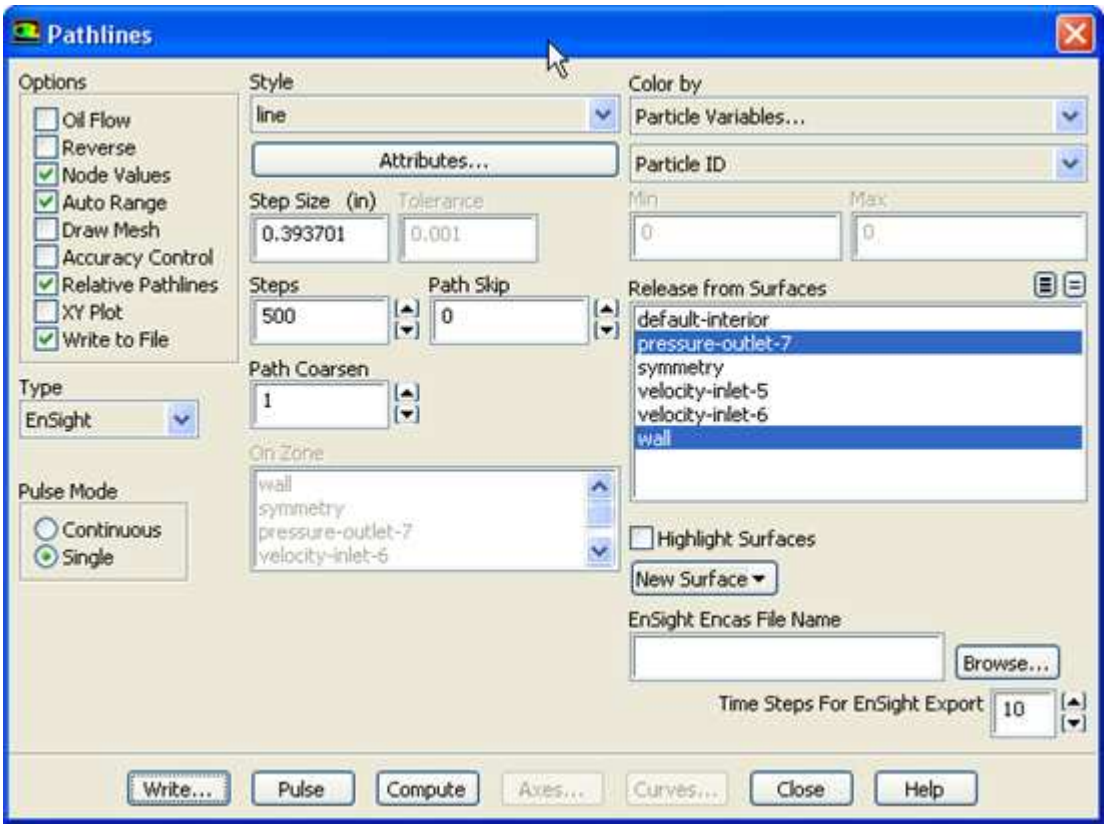
opens *The **Select File** Dialog Box* (p. 33), in which you can specify a file in which to save all of the custom vectors in the **Vectors** list.

Load...

opens the **Select File** dialog box, in which you can specify a file from which to read custom vectors (i.e., a file that you saved using the **Save...** button above).

35.19.7. Pathlines Dialog Box

The **Pathlines** dialog box controls the generation and display of pathlines. See *Displaying Pathlines* (p. 1400) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

contains the options described below:

Oil Flow

toggles between regular pathlines and oil-flow pathlines. When this option is selected, pathlines are constrained to lie on the zone selected in the **On Zone** list.

Reverse

specifies that each particle's path is traced back in time. This option is turned off by default.

Node Values

specifies that node values should be interpolated to compute the scalar field at a particle location. This option is turned on by default. See *Choosing Node or Cell Values* (p. 1409) for details.

Auto Range

specifies that the minimum and maximum values of the scalar field will be the limits of that field. If this option is not selected, you can enter **Min** and **Max** values manually. These values determine the range of the color scale.

Draw Mesh

when enabled opens the *Mesh Display Dialog Box* (p. 1635) where you can specify how to display your mesh.

Accuracy Control

allows you to specify tolerance to control the pathlines accuracy.

Relative Pathlines

allows you to display the pathlines relative to the rotating reference frame.

XY Plot

enables the display of an XY plot along the pathline trajectories. When this option is selected, the **Display** push button will change to **Plot**.

Write to File

activates the file-writing option. When this option is selected, the **Type** option becomes active and the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open *The **Select File** Dialog Box* (p. 33), in which you can specify a name and save a file containing the XY plot data. The format of this file is described in *XY Plot File Format* (p. 1477).

Type

specifies the type of the file you want to write.

CFD-Post

allows you to write the file in CFD-Post compatible format, which can be read in **CFD-Post**.

Fieldview

allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

Geometry

allows you to write the file in.ibl format, which can be read in GAMBIT.

EnSight

allows you to write the file in.encas format, which can be read in EnSight.

Pulse Mode

specifies either a single or continuous release of particles that follow the pathlines, starting at the release surface, when you use the **Pulse** button to animate the pathlines.

Continuous

sets the pulse mode to continuously release particles from the initial positions.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Single
sets the pulse mode to release a single wave of particles.

The **Pulse Mode** is used only when particles are being pulsed.

Style
sets the pathline style. Pathlines can be displayed as lines, ribbons, spheres, cylinders, or a set of points. The **ribbon** style also uses the **Twist By** field and the **TwistFactor** value (specified in the *Ribbon Attributes Dialog Box* (p. 1979)). Pulsing can be done only on **point** or **line** styles.

Attributes...
opens the *Path Style Attributes Dialog Box* (p. 1979) or the *Ribbon Attributes Dialog Box* (p. 1979), depending on the selected **Style**. These dialog boxes let you set the attributes (ribbon width, marker size, cylinder radius, twist-by field, etc.) for the chosen pathline style.

Step Size
sets the interval used for computing the next position of a particle. This is in units of length. Particle positions are always computed when they enter/leave a cell. If you specify a very large size, the particle positions at entry/exit of each cell will still be computed (and displayed).

Tolerance
allows you to control the error when using large time step sizes for the calculation. This is available when **Accuracy Control** is selected under **Options**.

Steps
sets the maximum number of steps a particle can advance. A particle's path may also end if it leaves the domain.

Path Skip
allows you to "thin" or "sample" the pathlines that are displayed. See *"Thinning" Pathlines* (p. 1404) for details.

Path Coarsen
reduces the time and the number of points plotted in a pathline. Coarsening factor of 'n' will result in plotting of each 'n'th point for a given pathline in each cell.

On Zone
contains a list from which you can select the zone on which the particles are constrained to lie. This list is activated only when the **Oil Flow** option is selected.

Color by
contains a list from which you can select the scalar field to be used to color the pathlines. The default is **Particle ID**, a unique ID for each particle.

Y Axis Function
contains a list of solution variables that can be used for the y axis of the plot.

This item appears when the **XY Plot** option is turned on.

X Axis Function
contains a list of functions for the x axis of the plot. You can plot the quantity selected in the **Y Axis Function** drop-down list as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory.

This item appears when the **XY Plot** option is turned on.

Min/Max
display the values to which the lower and upper ends of the color scale map. If you are using the **Auto Range** option, these values will be automatically computed when you press **Compute**. If you are not using **Auto Range**, you can enter values manually.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release from Surfaces
selects the surfaces from which to release particles. You must select at least one surface before any particles can be released.

Highlight Surfaces
when enabled highlights the surfaces (selected in the **Pathlines** dialog box) in the graphics window.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the *Point Surface Dialog Box* (p. 1927).

Line/Rake
opens the *Line/Rake Surface Dialog Box* (p. 1928).

Plane
opens the *Plane Surface Dialog Box* (p. 1929).

Quadric
opens the *Quadric Surface Dialog Box* (p. 1931).

Iso-Surface
opens the *Iso-Surface Dialog Box* (p. 1933).

Iso-Clip
opens the *Iso-Clip Dialog Box* (p. 1935).

EnSight Encas File Name
allows you to specify a.encas file name if you selected **EnSight** under **Type**.

Browse...
opens the *The **Select File** Dialog Box* (p. 33) where you can select the `.encas` file.

Time Steps For EnSight Export
allows you to set the number of time levels that are available for animation in EnSight.

Display
displays the pathlines and records all dialog box settings.

Plot
displays an XY plot along the pathline trajectories. This button replaces the **Display** button when the **XY Plot** option is turned on.

Write...
opens *The **Select File** Dialog Box* (p. 33), in which you can save the XY plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Pulse
animates the position of particles. If **Pulse Mode** is set to **Continuous**, particles are reintroduced at the seed positions repeatedly. If the mode is **Single**, a single wave of particles moves through the domain.

The label of this button changes to **Stop !** when the animation is in progress. Press **Stop !** to stop pulsing (and the label goes back to **Pulse**).

Compute
computes the **Min** and **Max** for the field chosen in **Color by**.

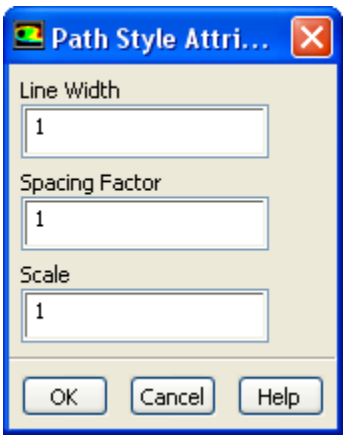
Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the XY plot axes. This item appears when the **XY Plot** option is turned on.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot. This item appears when the **XY Plot** option is turned on.

35.19.8. Path Style Attributes Dialog Box

To modify the line width, cylinder radius or marker size, use the **Path Style Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the *Pathlines Dialog Box* (p. 1975). See *Controlling the Pathline Style* (p. 1403) for details about the items below.



Controls

Line Width/Marker Size/Width/
determines the thickness of the pathlines.

Diameter
specifies the diameter of the sphere. This parameter appears only when **sphere** is selected under **Style** in the **Pathlines** dialog box.

Spacing Factor
controls the spacing of arrows when you use the **line-arrows** style.

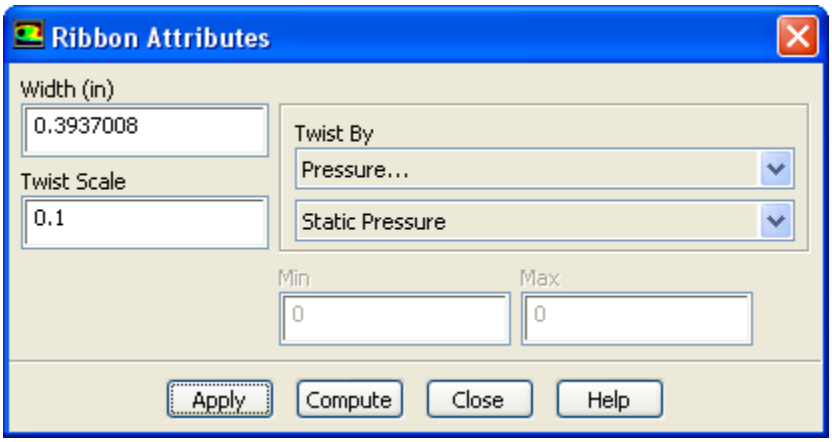
Scale
controls the size of the arrow heads when you use the **line-arrows** style.

Detail
specifies the detail applied to the graphical rendering of the spheres. This parameter appears only when **sphere** is selected under **Style** in the **Pathlines** dialog box.

35.19.9. Ribbon Attributes Dialog Box

To modify the ribbon width and set the scalar field by which to twist the ribbon, use the **Ribbon Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the *Pathlines Dialog Box* (p. 1975) when the selected **Style** is ribbon. See *Controlling the Pathline Style* (p. 1403) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Width
determines the thickness of the ribbon.

Twist Scale
sets the amount of twist for a given field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

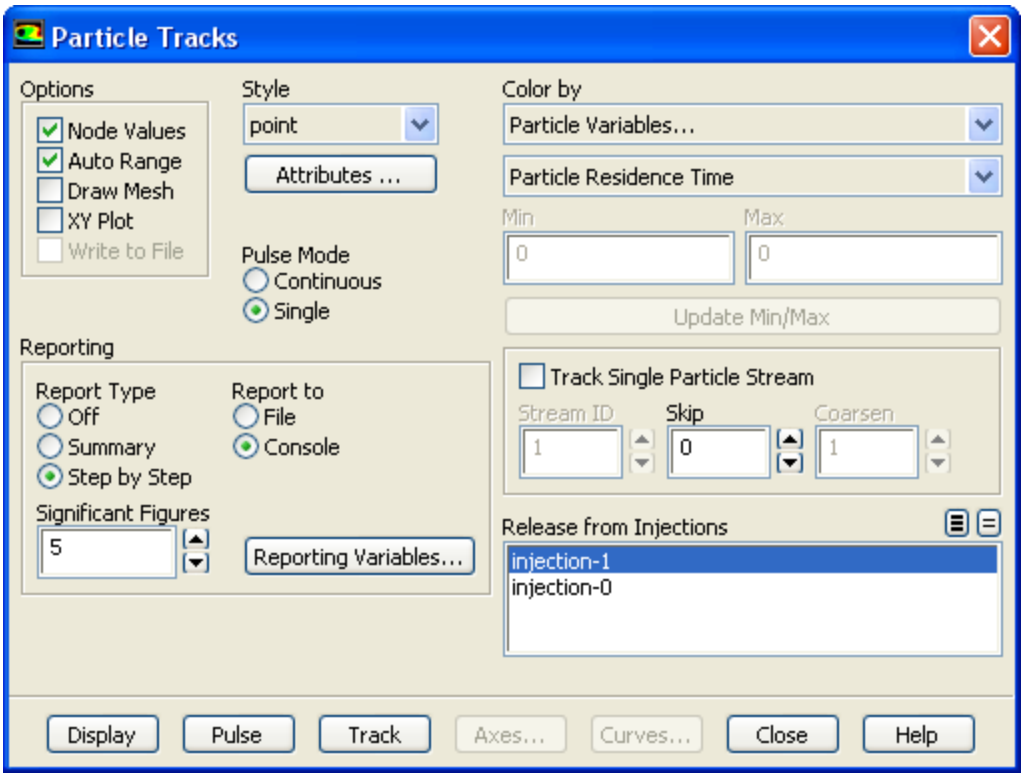
Twist By
contains a drop-down list from which you can select a scalar field on which pathline twisting is based.

Min/Max
displays the minimum/maximum value of the scalar field selected in **Twist By**.

35.19.10. Particle Tracks Dialog Box

The **Particle Tracks** dialog box controls the generation and display of discrete phase particles. See *Displaying of Trajectories* (p. 1068) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the options described below:

Node Values

specifies that node values should be interpolated to compute the scalar field at a particle location. This option is turned on by default. See *Choosing Node or Cell Values* (p. 1409) for details.

Auto Range

specifies that the minimum and maximum values of the scalar field will be the limits of that field. If this option is not selected, you can enter **Min** and **Max** values manually. These values determine the range of the color scale.

Draw Mesh

toggles between displaying and not displaying the mesh. The *Mesh Display Dialog Box* (p. 1635) is opened when **Draw Mesh** is selected.

XY Plot

enables the display of an XY plot along the particle trajectories. When this option is selected, the **Display** push button will change to **Plot**.

Write to File

(available only when **XY Plot** is on) activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open *The **Select File** Dialog Box* (p. 33), in which you can specify a name and save a file containing the XY plot data. The format of this file is described in *XY Plot File Format* (p. 1477).

Reporting

allows you to specify how you would like to report particle tracks.

Report Type

controls the type of trajectory-fate reports to be displayed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Off
disables reporting of trajectory fates.

Summary
enables summary reports of trajectory fates. See *Reporting of Trajectory Fates* (p. 1072) for details.

Step by Step
enables the step-by-step reporting of trajectories. This item will not appear if you have requested unsteady tracking in the *Discrete Phase Model Dialog Box* (p. 1723). See *Step-by-Step Reporting of Trajectories* (p. 1078) for details.

Current Positions
enables the reporting of the positions and velocities of all particles that are in the domain at the current time. This item appears only when unsteady tracking has been requested in the *Discrete Phase Model Dialog Box* (p. 1723). See *Reporting of Current Positions for Unsteady Tracking* (p. 1080) for details.

Report to
indicates the destination of the trajectory report.

File
enables the writing of the trajectory report to a file. When this option is enabled, the **Track** button will become the **Write...** button.

Console
enables the display of the trajectory report information in the console window.

Significant Figures
controls the number of significant figures used when a **Step by Step** or **Current Positions** report is selected.

Reporting Variables...
opens the *Reporting Variables Dialog Box* (p. 1984) that contains a list from which you can select the scalar field to be used to color the particle tracks.

Style
sets the trajectory style. Trajectories can be displayed as lines, ribbons, spheres, cylinders, or a set of points. The **ribbon** style also uses the **Twist By** field and the **Twist Scale** value (specified in the *Ribbon Attributes Dialog Box* (p. 1979)). Pulsing can be done only on **point** or **line** styles.

Attributes...
opens the *Path Style Attributes Dialog Box* (p. 1979) or the *Ribbon Attributes Dialog Box* (p. 1979), depending on the selected **Style**. These dialog boxes let you set the attributes (ribbon width, marker size, cylinder radius, twist-by field, etc.) for the chosen trajectory style.

Pulse Mode
specifies either a single or continuous release of particles that follow the trajectories, starting at the release surface, when you use the **Pulse** button to animate the trajectories.

Continuous
sets the pulse mode to continuously release particles from the initial positions.

Single
sets the pulse mode to release a single wave of particles.

The **Pulse Mode** is used only when particles are being **Pulse**d.

Y Axis Function
contains a list of solution variables that can be used for the y axis of the plot.

This item appears when the **XY Plot** option is turned on.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Axis Function

contains a list of functions for the x axis of the plot. You can plot the quantity selected in the **Y Axis Function** drop-down list as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory.

This item appears when the **XY Plot** option is turned on.

Min/Max

display the values to which the lower and upper ends of the color scale map. If you are using the **Auto Range** option, these values will be automatically computed when you press **Compute**. If you are not using **Auto Range**, you can enter values manually.

Update Min/Max

computes the **Min** and **Max** for the field chosen in **Color by**.

Track PDF Transport Particles

enables the tracking of PDF transport particles. This item is available only when the composition PDF transport model is enabled.

Track Single Particle Stream

enables the tracking of a single particle stream in the selected injection, instead of all the streams in that injection. See *Specifying Individual Particles for Display* (p. 1070) for details.

Stream ID

specifies the particle stream to be tracked when the **Track Single Particle Stream** option is enabled.

Skip

allows you to "thin" or "sample" the particles that are displayed when the **Track PDF Transport Particles** option is enabled.

Coarsen

reduces the plotting time by reducing the number of points that are plotted for a given trajectory in any cell. This is only valid for steady-state cases

Release from Injections

selects the injections from which to release particles. You must select at least one injection before any particles can be released.

Display

displays the trajectories and generates a trajectory report in the console window (if requested).

Plot

displays an XY plot along the particle trajectories. This button replaces the **Display** button when the **XY Plot** option is turned on.

Write...

opens *The **Select File** Dialog Box* (p. 33), in which you can save the XY plot data or trajectory report to a file. This button replaces the **Plot** button when the **Write to File** option is turned on, or the **Track** button when the **File** option is selected under **Report to**.

Pulse

animates the position of particles. If **Pulse Mode** is set to **Continuous**, particles are reintroduced at the seed positions repeatedly. If the mode is **Single**, a single wave of particles moves through the domain.

The label of this button changes to **Stop !** when the animation is in progress. Press **Stop !** to stop pulsing (and the label goes back to **Pulse**).

Track

computes the particle trajectories and generates any reports without displaying the trajectories in the graphics window.

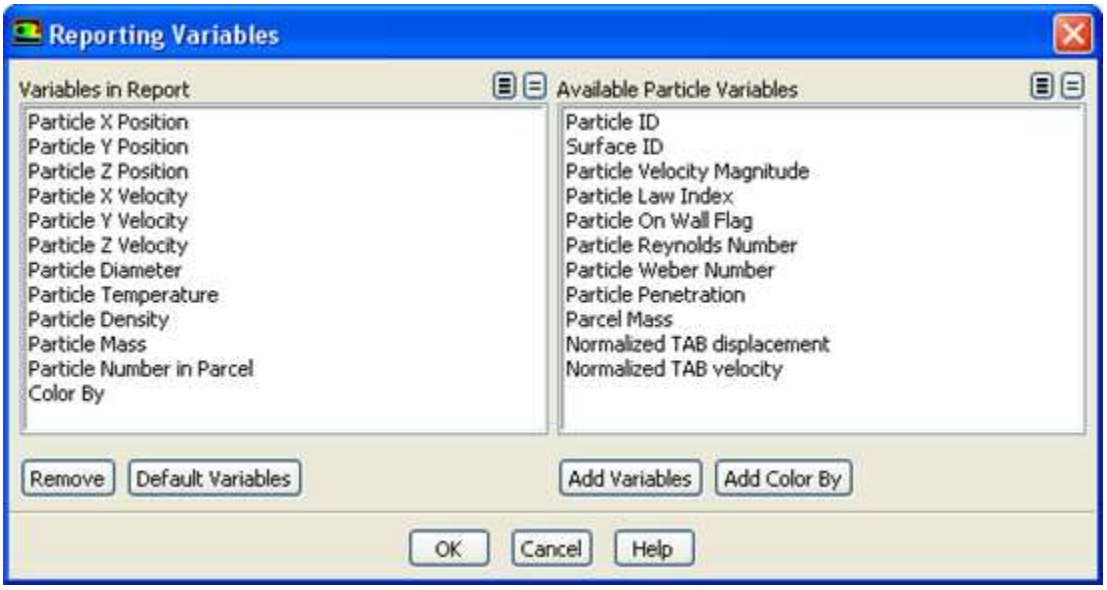
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the XY plot axes. This item appears when the **XY Plot** option is turned on.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot. This item appears when the **XY Plot** option is turned on.

35.19.11. Reporting Variables Dialog Box

The **Reporting Variables** dialog box allows you to control the particle variables that you include in your reporting. See *Step-by-Step Reporting of Trajectories* (p. 1078) for details about the items below.



Controls

Variables in Report
contains all variables currently in the report.

Remove
removes the selected variable from the report.

Default Variables
restores the default list.

Available Particle Variables
contains the particle variables that are available for you to select.

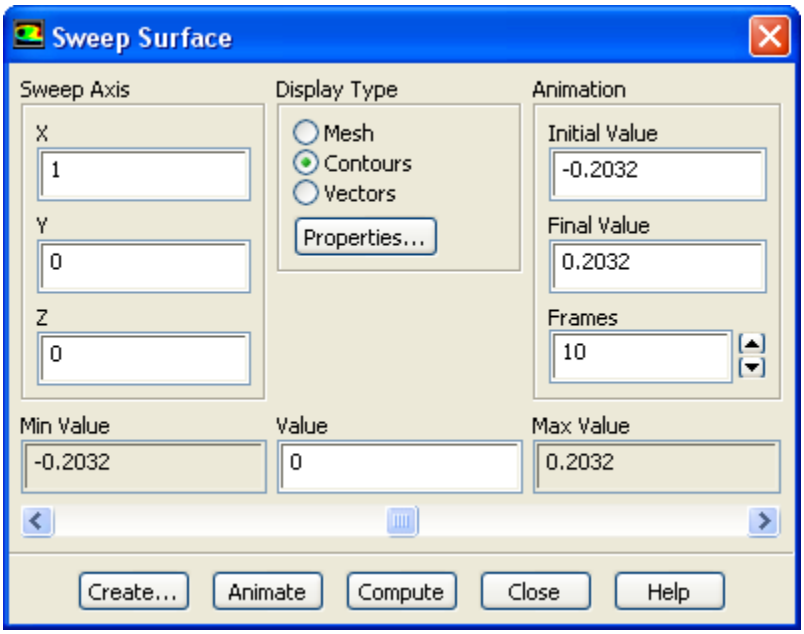
Add Variables
takes the selected variable from the **Available Particle Variables** list and adds it to the **Variables in Report** list.

Add Color By
adds the **Color by** variable to the **Variables in Report** list.

35.19.12. Sweep Surface Dialog Box

The **Sweep Surface** dialog box controls the display and animation of mesh, contour, and vector plots generated on a sweep surface. See *Displaying Results on a Sweep Surface* (p. 1409) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Sweep Axis
specifies the (**X**, **Y**, **Z**) vector representing the axis along which the surface should be swept.

Display Type
specifies the type of display to be swept through the domain (**Mesh**, **Contours**, or **Vectors**).

Properties...
opens the *Contours Dialog Box* (p. 1967) if **Contours** is the selected **Display Type**, or the *Vectors Dialog Box* (p. 1970) if **Vectors** is the selected **Display Type**. (This button is not available if **Mesh** is the selected **Display Type**.)

Animation
contains controls for animating the sweep-surface display.

Initial Value, Final Value
specify the initial and final positions for the animation.

Frames
specifies the number of frames in the animation.

Min Value, Max Value
show the minimum and maximum extents of the domain along the specified **Sweep Axis**. These values are updated when you click **Compute**.

Value
shows the current position at which the requested display is plotted. You can change the value by moving the slide bar below it, or by entering a new value and pressing the <RETURN> key.

Create...
opens the *Create Surface Dialog Box* (p. 1986), where you can create a surface from the currently-displayed sweep surface.

Animate
animates the display, sweeping the requested display through the domain along the specified axis.

Compute
updates the **Min Value** and **Max Value** to reflect the minimum and maximum extents of the domain along the specified **Sweep Axis**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.19.13. Create Surface Dialog Box

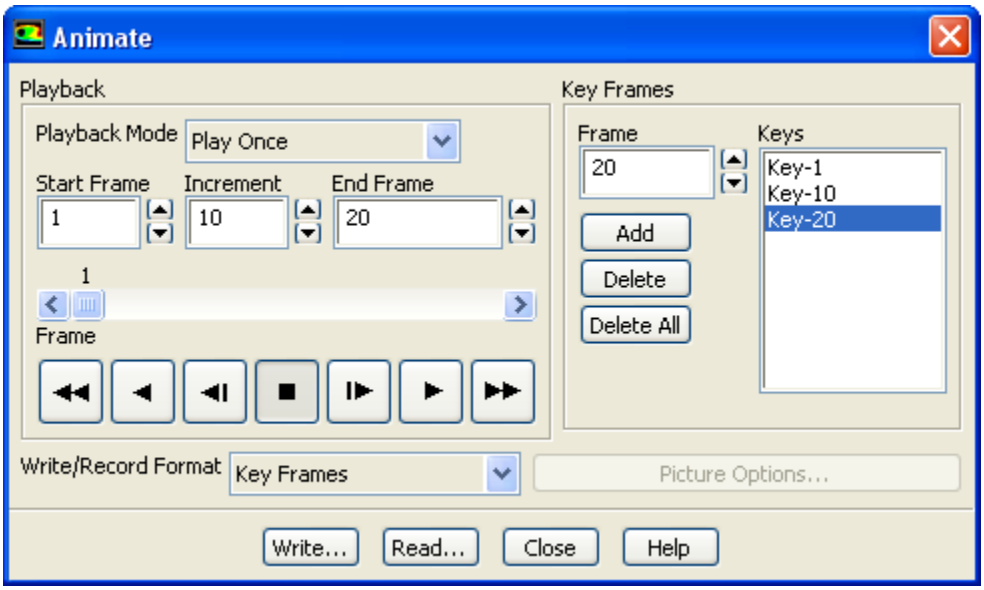
The **Create Surface** dialog box allows you to save a sweep surface for later use. You can open it by clicking **Create...** in the *Sweep Surface Dialog Box* (p. 1984). See *Displaying Results on a Sweep Surface* (p. 1409) for details about the items below.

Controls

Surface Name
specifies a name for the surface to be created.

35.19.14. Animate Dialog Box

The **Animate** dialog box allows you to specify key frames that define the basic movement of an animated sequence, and then play back the animation. ANSYS FLUENT interpolates between your specified key frames. See *Animating Graphics* (p. 1454) for details about the items below.



Controls

Playback
contains the controls that you use to play back the animation. See *Playing an Animation* (p. 1455) for details.

Playback Mode
contains a drop-down list of playback options.

Play Once
sets the option to play back frames from **Start Frame** to **End Frame** once.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Repeat
sets the option to continually play back frames from **Start Frame** to **End Frame**.

Auto Reverse
sets the option to continually play back the images while reversing playback direction after each set.

Start Frame, End Frame
set the frames at which the animation should begin and end. By changing these numbers you can view a subset of the frames.

Increment
sets the number of frames to increment the frame-counter by when you use the fast-forward or fast-reverse buttons.

Frame
shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If you slide the bar to a different location, the frame corresponding to the new frame number will be displayed in the graphics window.

(Tape Player Buttons)
allow you to play the animation forward and backward, fast-reverse and fast-forward the animation, and stop it. The buttons function in a way similar to those on a standard video cassette player.

Key Frames
contains the controls that you use to define the key frames for the animation. See *Creating an Animation* (p. 1455) for details.

Frame
sets the number to be assigned to the next key frame added to the list of **Keys**.

Keys
contains a list of the key frames that have been defined. If you select a key frame in this list, the associated scene will be displayed in the graphics window.

Add
creates a key frame with the number shown in **Frame** for the scene currently displayed in the graphics window.

Important

Be sure to change the frame number before you add the new key frame so that you will not overwrite the last key frame that you created.

Delete
deletes the key frame that is selected in the **Keys** list.

Delete All
deletes all key frames in the **Keys** list.

Write/Record Format
specifies **Key Frames**, **Picture Files**, **MPEG**, or **Video** (not available on Windows) as the format in which to save the animation. See *Saving an Animation* (p. 1456) for details about these options.

Picture Options...
opens the *Save Picture Dialog Box* (p. 1988), in which you can specify parameters for saving the animation to picture files. This button is available only when **Picture Files** is selected as the **Write/Record Format**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write...

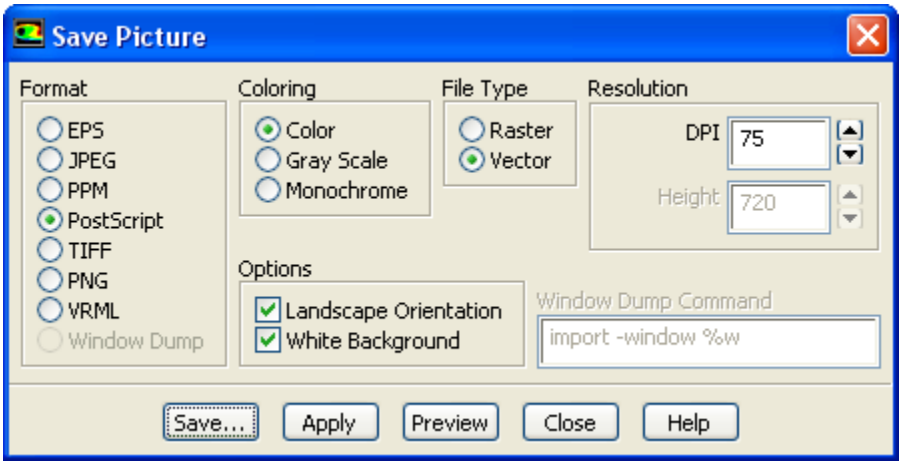
opens *The **Select File** Dialog Box* (p. 33), in which you can specify a name for the animation file and save it.

Read...

opens the **Select File** dialog box, in which you can specify the name of the animation file to be read. Note that the current case and data should contain the surfaces and any other information that the key frame description refers to. See *Reading an Animation File* (p. 1458).

35.19.15. Save Picture Dialog Box

The **Save Picture** dialog box allows you to set save picture parameters and save picture files of graphics windows. See *Saving Picture Files* (p. 117) for details on the use of this dialog.



Controls

Format

allows you to select the format of picture files.

EPS

(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs which import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

JPEG

is a common raster file format.

PPM

output is a common raster file format.

PostScript

is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF

is a common raster file format.

PNG

is a common raster file format.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VRML
is a graphics interchange format that allows export of 3D geometrical entities that you can display in the ANSYS FLUENT graphics window. This format can commonly be used by VR systems and in particular the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in ANSYS FLUENT, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump
(UNIX systems only) selects a window dump operation for generating the picture. With this format, you will need to specify the appropriate **Window Dump Command**.

Coloring
(all formats except **Window Dump**) specifies the color mode for the picture file.

Color
specifies a color-scale copy.

Gray Scale
specifies a gray-scale copy.

Monochrome
specifies a black-and-white copy.

Important

Most monochrome PostScript devices will render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray Scale**.

File Type
specifies the type of picture file to be saved. See *Choosing the File Type* (p. 120) for details.

Raster
specifies a raster type picture. The supported raster formats are **EPS**, **JPEG**, **PPM**, **PostScript**, **TIFF**, and **PNG**.

Vector
specifies a vector type picture. The supported vector formats are **EPS**, **PostScript**, and **VRML**.

Resolution
specifies the resolution or the size (in pixels) of the picture.

DPI
specifies the resolution of **EPS** and **PostScript** files in dots per inch (DPI). The default value for **DPI** is set to 75.

Width
specifies the width of the raster picture image.

Height
specifies the height of the raster picture image.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*The default value for **Width** and **Height** is set to zero, so that the default picture is generated at the same resolution as the active graphics window.*

Options

contains additional options for all picture formats except **Window Dump**.

Landscape Orientation

specifies the orientation of the picture. If this option is enabled, the picture is made in landscape mode; otherwise, it is made in portrait mode.

White Background

controls the foreground/background color. If this option is enabled, the foreground and background colors of graphics windows being saved as pictures will be swapped. Hence, it allows you to save pictures with a white background and a black foreground, while the graphics windows are displayed with a black background and white foreground.

Window Dump Command

(UNIX systems only) specifies the command to be used to save the picture file, when you select the **Window Dump** format. See *Window Dumps (Linux/UNIX Systems Only)* (p. 121) for details.

Save...

opens *The **Select File** Dialog Box* (p. 33), in which you can specify a name for the picture file to be saved and then save the file. The resulting file will contain a picture of the active graphics window.

Apply

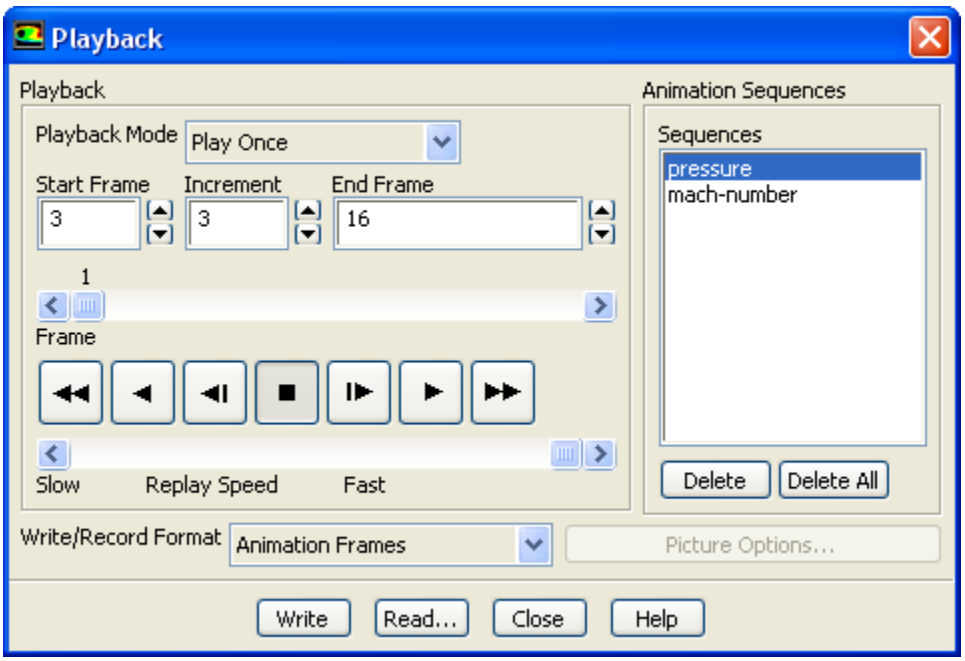
saves the current settings. ANSYS FLUENT will use these settings when making subsequent pictures.

Preview

applies the current settings to the active graphics window so that you can investigate the effects of different options interactively before saving the final picture.

35.19.16. Playback Dialog Box

The **Playback** dialog box allows you to play back an animation sequence. See *Playing an Animation Sequence* (p. 1300) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Playback
contains the controls that you use to play back the selected animation sequence.

Playback Mode
contains a drop-down list of playback options.

Play Once
sets the option to play back frames from **Start Frame** to **End Frame** once.

Auto Repeat
sets the option to continually play back frames from **Start Frame** to **End Frame**.

Auto Reverse
sets the option to continually play back the images while reversing playback direction after each set.

Start Frame, End Frame
set the frames at which the animation should begin and end. By changing these numbers you can view a subset of the frames.

Increment
sets the number of frames to increment the frame-counter by when you use the fast-forward or fast-reverse buttons.

Frame
shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If you slide the bar to a different location, the frame corresponding to the new frame number will be displayed in the graphics window.

(Tape Player Buttons)
allow you to play the animation forward and backward, fast-reverse and fast-forward the animation, and stop it. The buttons function in a way similar to those on a standard video cassette player.

Replay Speed
controls the playback speed for the animation. Move the **Replay Speed** slider bar to the left to reduce the playback speed (and to the right to increase it).

Animation Sequences
contains the controls that you use to define the sequence to be played back.

Sequences
contains a list of the animation sequences that have been defined.

Delete
deletes the animation sequence that is selected in the **Sequences** list.

Delete All
deletes all animation sequences in the **Sequences** list.

Write/Record Format
specifies **Animation Frames**, **Picture Files**, or **MPEG** as the format in which to save the animation. See *Saving an Animation Sequence* (p. 1302) for details about these options.

Picture Options...
opens the *Save Picture Dialog Box* (p. 1988), in which you can specify parameters for saving the animation to picture files. This button is available only when **Picture Files** is selected as the **Write/Record Format**.

Write
saves the specified file(s) in the current working directory.

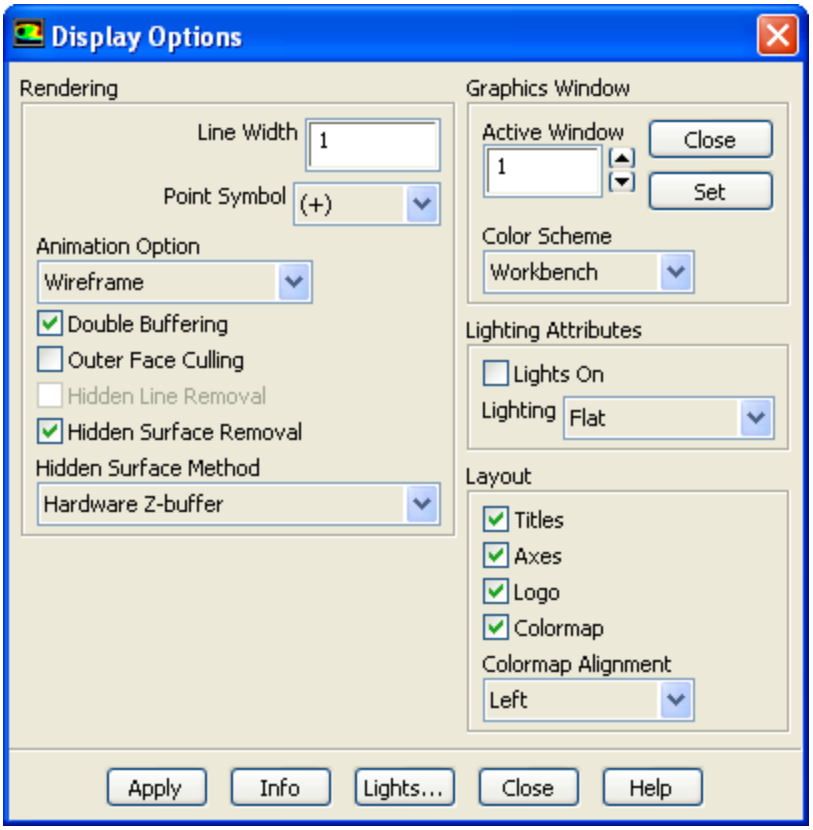
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read...

opens *The **Select File** Dialog Box* (p. 33), in which you can specify the name of the solution animation file to be read. See *Reading an Animation Sequence* (p. 1303) for details.

35.19.17. Display Options Dialog Box

The **Display Options** dialog box provides an interactive mechanism for setting attributes or options that control how and where a scene is rendered.



Controls

Rendering

allows you to modify characteristics of the display that are related to the way in which scenes are rendered. See *Modifying the Rendering Options* (p. 1425) for details about these items.

Line Width

controls the thickness of lines. The default is 1.

Point Symbol

sets the symbol used for nodes and data points.

Animation Option

contains a drop-down list of animation options: **All** and **Wireframe**. **Wireframe** uses a wireframe representation of all geometry during mouse manipulation This option is turned on by default. You should turn it off only if your computer has a graphics accelerator; otherwise the mouse manipulation may be very slow.

Double Buffering

turns double buffering on or off, if it is supported by the driver. Double buffering dramatically reduces screen flicker during graphics updates. Note that if your display hardware does not support double

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Outer Face Culling

allows you to turn off the display of outer faces in wall zones. **Outer Face Culling** is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you turn on the **Outer Face Culling** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (i.e., walls with fluid or solid cells on both sides).

Hidden Line Removal

turns hidden line removal on or off. If you do not use hidden line removal, ANSYS FLUENT will not try to determine which lines in the display are behind others; it will display all of them, and a cluttered display will result. You should turn this option off if you are working with a 2D problem or with geometries that do not overlap. Note that this option is not available when using the **Workbench Color Scheme**.

Hidden Surface Removal

turns hidden surface removal on or off. If you do not use hidden surface removal, ANSYS FLUENT will not try to determine which surfaces in the display are behind others; it will display all of them, and a cluttered display will result. You should turn this option off if you are working with a 2D problem or with geometries that do not overlap.

Hidden Surface Method

chooses the method to be used for hidden surface removal. These options vary in speed and quality, depending on the device you are using. The choices are listed below.

Hardware Z-buffer

is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware dependent.

Painters

will show fewer edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

Software Z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

Z-sort only

is a fast software method, but it is not as accurate as software-z-buffer.

Graphics Window

allows you to open and close graphics windows. See *Opening Multiple Graphics Windows* (p. 1412) for details.

Active Window

indicates the graphics window to be opened, closed, or set active. (A graphics window's ID is displayed in its title.)

Open/Close

opens or closes the window with the ID shown in the **Active Window** box. If the indicated window is open, the **Close** button will appear, and if the indicated window is not open, the **Open** button will appear.

Set

sets the window with the ID shown in the **Active Window** box to be the active graphics window. See *Setting the Active Window* (p. 1413) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color Scheme
contains a drop-down list of available graphics window color schemes to choose from. Choices are **Workbench** (blue background) and **Classic** (black background).

Lighting Attributes
controls lighting attributes for all lights in the active graphics window. See *Adding Lights* (p. 1423) for details.

Lights On
turns *all* lights in the active graphics window on or off.

Lighting
specifies the method to be used in lighting interpolation: **Flat**, **Gouraud**, or **Phong**. (**Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.)

Layout
controls the display of captions, axes and the colormap in the graphics display window. See *Changing the Legend Display* (p. 1413) for details.

Titles
enables/disables the display of all captions in the graphics display window.

Axes
enables/disables the display of the axis triad.

Logo
allows you to hide or display the ANSYS logo.

Colormap
enables/disables the display of the color scale.

Colormap Alignment
allows you to adjust the alignment of colormap in the graphics display. Select the side (i.e. **Top**, **Bottom**, **Left**, and **Right**) from the drop-down list, where you want to align the colormap.

Apply
applies the specified attributes and re-renders the scene in the active graphics window with the new attributes. To see the effect of the new attributes on other graphics windows, you must redisplay them.

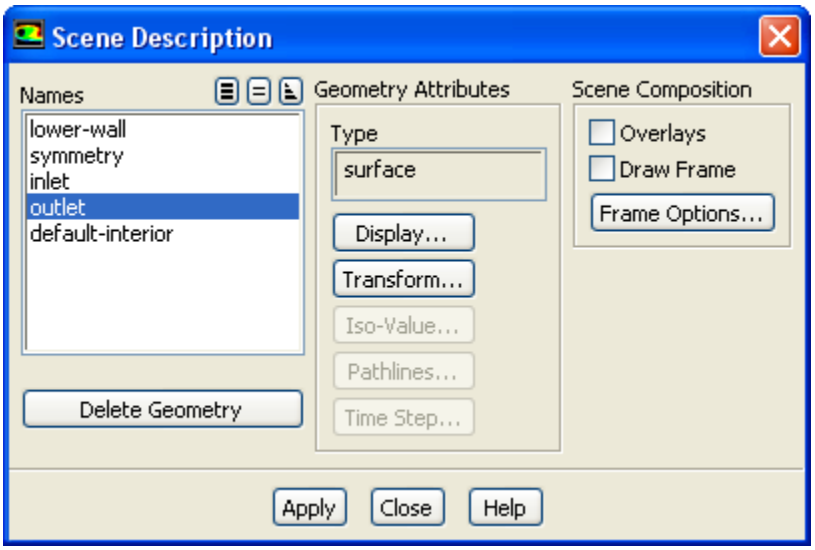
Info
prints out information about your graphics driver in the console.

Lights...
opens the *Lights Dialog Box* (p. 2005), which allows you to create, delete, and modify directional light sources.

35.19.18. Scene Description Dialog Box

The **Scene Description** dialog box allows you to turn overlays on and off, select geometric objects in the display for modification or deletion, and open dialog boxes that control various characteristics of the selected object(s). (Note that you cannot use the **Scene Description** dialog box to control XY plot and histogram displays.) See *Composing a Scene* (p. 1444) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Names

contains a list of the geometric objects that currently exist in the scene (including those that are presently invisible). You can specify the object or objects to be manipulated by selecting names in this list. If you select more than one object at a time, any operation (transformation, color specification, etc.) will apply to all the selected objects. You can also select objects by clicking on them in the graphics display using the mouse probe button, which is, by default, the right mouse button. (See *Controlling the Mouse Button Functions* (p. 1427) for information about mouse button functions.) To deselect a selected object, simply click on its name in the **Names** list. See *Selecting the Object(s) to be Manipulated* (p. 1445)for details.

Geometry Attributes

contains information about the type of the selected geometric object and push buttons which open dialog boxes for modifying the object.

Type

reports the type of the selected object. Possible types include `mesh`, `surface`, `contour`, `vector`, and `Group`. This information is especially helpful when you need to distinguish two or more objects with the same name. When more than one object is selected, the type displayed is `Group`.

Display...

opens the *Display Properties Dialog Box* (p. 1996), which allows you to change the color, visibility, and other properties for the selected object.

Transform...

opens the *Transformations Dialog Box* (p. 1997), which allows you to translate, rotate, and scale the selected object.

Iso-Value...

opens the *Iso-Value Dialog Box* (p. 1998), which allows you to change the isovalue of an isosurface. This push button is available only if the geometric object selected in the **Names** list is an isosurface or an object on an isosurface (contour on an isosurface, for example); otherwise it is grayed out.

Pathlines...

opens the *Pathline Attributes Dialog Box* (p. 1999), which allows you to set the maximum number of steps for the selected pathlines. This is most useful in animating the path of a set of particles.

Scene Composition

contains controls for enabling overlays and bounding frames.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overlays
activates the superimposition of a new geometry onto a currently displayed geometry. See *Overlay of Graphics* (p. 1411) for details.

Draw Frame
activates the display of a bounding frame in the graphics display. See *Adding a Bounding Frame* (p. 1452) for details.

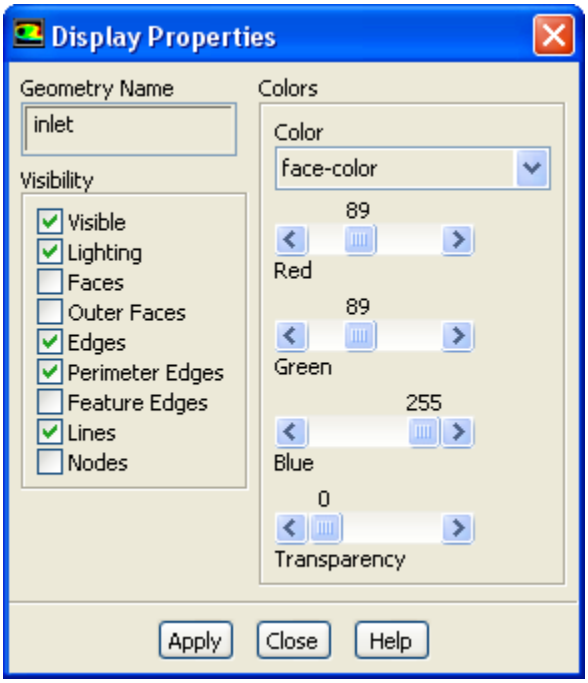
Frame Options...
opens the *Bounding Frame Dialog Box* (p. 2000), in which you can define properties of the bounding frame. See *Adding a Bounding Frame* (p. 1452) for details.

Delete Geometry
deletes the geometric object that is currently selected in the **Names** list. The ability to delete individual objects is especially useful if you have overlays on and you generate an unwanted object (e.g., if you generate contours of the wrong variable). You can simply delete the unwanted object and continue your scene composition, instead of starting over from the beginning.

Apply
saves the status of **Overlays** and **Draw Frame**. When you turn **Overlays** or **Draw Frame** on or off, you must click the **Apply** button to see the effect of the change on subsequent display operations.

35.19.19. Display Properties Dialog Box

To modify the color, visibility, and other display properties for individual geometric objects in the graphics display, use the **Display Properties** dialog box. You can open this dialog box by clicking the **Display...** push button in the *Scene Description Dialog Box* (p. 1994). See *Changing an Object's Display Properties* (p. 1446) for details about the items below.



Controls

Geometry Name
displays the name of the object you selected for modification in the *Scene Description Dialog Box* (p. 1994).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visibility

contains check buttons that control options related to the visibility of the selected object. See *Controlling Visibility* (p. 1446) for details.

Visible

toggles the visibility of the selected object. If it is turned on, the object will be visible in the display, and if it is turned off, the object will be invisible.

Lighting

turns the effect of lighting for the selected object on or off. You can choose to have lighting affect only certain objects instead of all of them. Note that if **Lighting** is turned on for an object such as a contour or vector plot, the colors in the plot will not be exactly the same as those in the colormap at the left of the display.

Faces

toggles the filled display of faces for the selected mesh or surface object. Turning **Faces** on has the same effect as turning on the display of faces in the *Mesh Display Dialog Box* (p. 1635).

Outer Faces

toggles the display of outer faces.

Edges

toggles the display of interior and exterior edges of the geometric object.

Perimeter Edges

toggles the display of the outline of the geometric object. (This option has no effect on the display of meshes.)

Feature Edges

toggles the display of feature lines (if any) of the geometric object.

Lines

toggles the display of the lines (if any) in the geometric object.

Nodes

toggles the display of the nodes (if any) in the geometric object.

Colors

contains controls for setting face, edge, line, and node colors, and transparency for faces. See *Controlling Object Color and Transparency* (p. 1447) for details.

Color

specifies the face, edge, line, or node color for modification. When you turn on this button, the color scales below will show the current color specification, which you can modify by moving the sliders on the color scales.

Red, Green, Blue

are color scales with which you can specify the RGB components of the face, edge or line color.

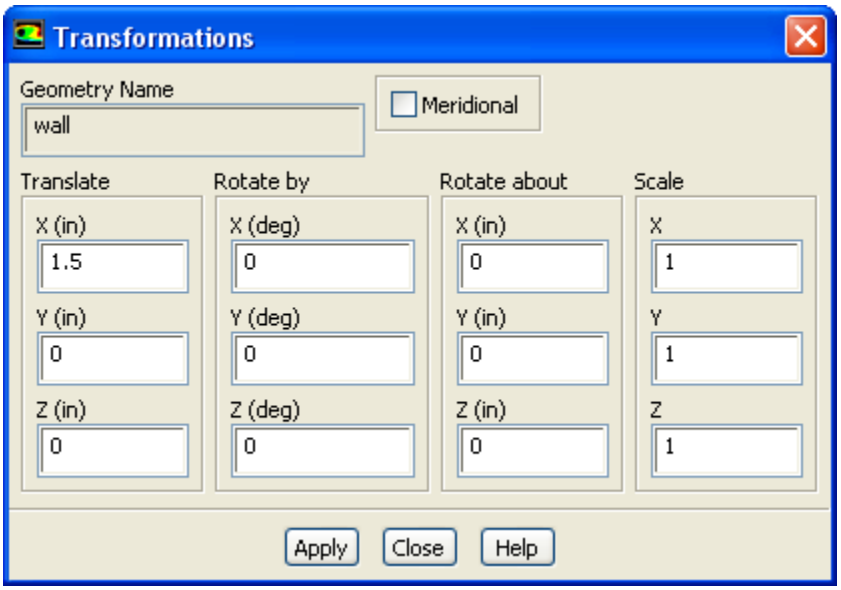
Transparency

sets the relative transparency of the selected object. An object with a transparency of 0 is opaque, and an object with a transparency of 100 is transparent.

35.19.20. Transformations Dialog Box

You can use the **Transformations** dialog box to translate, rotate, or scale individual objects in the graphics display. To open this dialog box, click the **Transform...** push button in the *Scene Description Dialog Box* (p. 1994). See *Transforming Geometric Objects in a Scene* (p. 1448) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Geometry Name
displays the name of the object you selected for modification in the *Scene Description Dialog Box* (p. 1994).

Meridional
(3D only) enables the display of the meridional view. This option is especially useful in turbomachinery applications.

Translate
contains **X**, **Y**, and **Z** real number fields in which you can enter the distance by which to translate the selected object in each direction.

Rotate by
contains **X**, **Y**,and **Z** integer number fields in which you can enter the number of degrees by which to rotate the selected object about each axis.

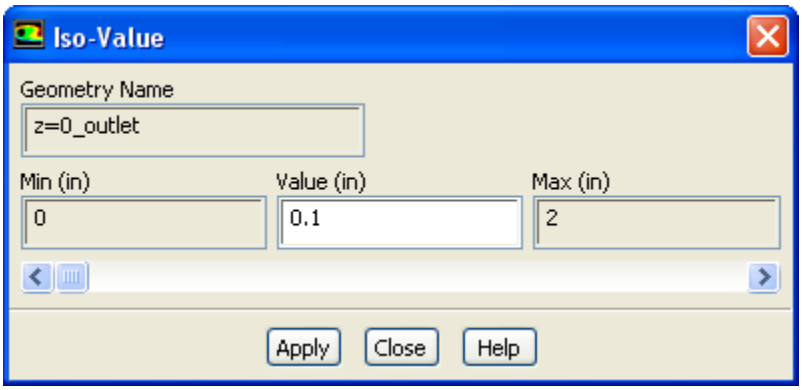
Rotate about
specifies the point about which to rotate the object.

Scale
contains **X**, **Y**, and **Z** real number fields in which you can enter the amount by which to scale the selected object in each direction. To avoid distortion of the object's shape, be sure to specify the same value for all three entries.

35.19.21. Iso-Value Dialog Box

The **Iso-Value** dialog box allows you to change the isovalue of an isosurface. The isosurface can be selected directly in the **Names** list or indirectly by selecting an object displayed on the isosurface. When you change the isovalue, any contours, vectors, etc. that were displayed on the original isosurface will be displayed on the isosurface with the new isovalue. To open this dialog box, click the **Iso-Value...** push button in the *Scene Description Dialog Box* (p. 1994). See *Modifying Iso-Values* (p. 1450) for details about using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

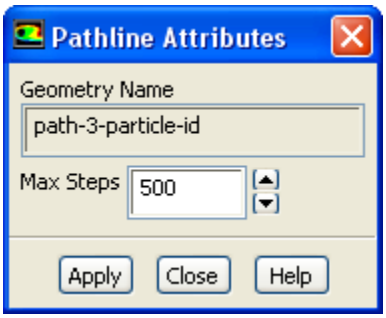
Geometry Name
displays the name of the geometric object (isosurface) you selected for modification in the *Scene Description Dialog Box* (p. 1994).

Min, Max
show the minimum and maximum values of the isosurface variable.

Value
sets the new isovalue for isosurfaces. After you change the value and click on **Apply**, contours, vectors, or pathlines that were displayed on the original isosurface will be displayed for the new isovalue. You can also use the slide bar to change the **Value**.

35.19.22. Pathline Attributes Dialog Box

The **Pathline Attributes** dialog box allows you to change the number of steps used in the computation of pathlines. This is most useful in creating animations of pathlines. To open this dialog box, click the **Pathlines...** button in the *Scene Description Dialog Box* (p. 1994). See *Modifying Pathline Attributes* (p. 1451) for details about using this dialog box.



Controls

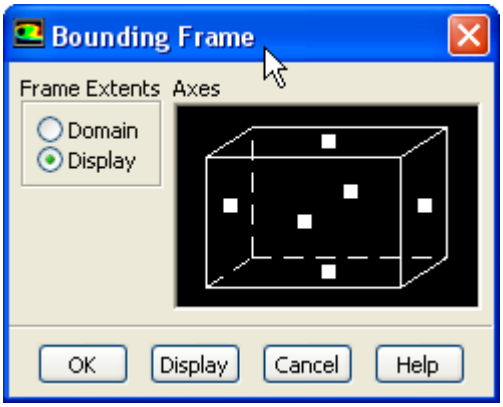
Geometry Name
displays the name of the geometric object you selected for modification in the *Scene Description Dialog Box* (p. 1994).

Max Steps
sets the new maximum number of steps for pathline computation. After you change the value and click on **Apply**, the selected pathline will be recomputed and redrawn.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.19.23. Bounding Frame Dialog Box

The **Bounding Frame** dialog box allows you to add a bounding frame with optional measure markings to the display of the domain. See *Adding a Bounding Frame* (p. 1452) for details about the items in this dialog box.



Controls

Frame Extents
indicates the extents of the bounding frame.

Domain
specifies that the frame should encompass the domain extents.

Display
specifies that the frame should encompass the portion of the domain that is shown in the display.

Axes
contains controls for specifying the frame boundaries and measurements. See *Adding a Bounding Frame* (p. 1452) for instructions on using these items.

Display
updates the active graphics window with the current frame settings.

35.19.24. Views Dialog Box

With the **Views** dialog box, you can make various modifications to the view displayed in the active graphics window. See *Modifying the View* (p. 1433) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Views

lists the currently defined views. Clicking on a view name highlights that name and enters it into the **Name** field. Double-clicking on a view name restores that view in the active graphics window.

Save Name

specifies the name to use when saving a view.

Actions

contains buttons for performing various actions related to the **Views** list and the **Save Name**.

Default

restores the "front" view in the active graphics window.

Auto Scale

modifies the view in the active graphics window by scaling and centering the current scene without changing its orientation.

Previous

allows you to return to previous displays.

Save

stores the view in the active graphics window with the name in the **Save Name** box. See *Saving Views* (p. 1440) for details.

Delete

removes the selected view name from the **Views** list.

Important

Be careful not to delete any of the pre-defined views.

Read...

opens *The **Select File** Dialog Box* (p. 33), in which you can specify the name of a view file to be read. See *Reading View Files* (p. 1440) for details.

Write...

opens the *Write Views Dialog Box* (p. 2002), in which you can select the views to be saved to a view file. See *Saving Views* (p. 1440) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mirror Planes

displays a list of all symmetry planes in the domain. Mirror images are drawn for all selected symmetry planes. See *Mirroring and Periodic Repeats* (p. 1441) for details.

Define Plane...

opens the *Mirror Planes Dialog Box* (p. 2002), in which you can define a mirror plane for a non-symmetric domain.

Periodic Repeats

indicates the number of periodic repetitions to be displayed. See *Mirroring and Periodic Repeats* (p. 1441) for details.

Define...

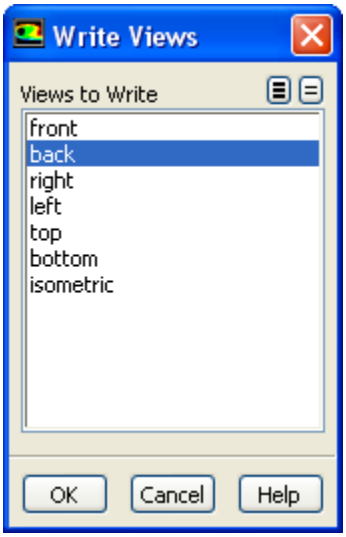
opens the *Graphics Periodicity Dialog Box* (p. 2003), in which you can define periodicity for a periodic domain.

Camera...

opens the *Camera Parameters Dialog Box* (p. 2005).

35.19.25. Write Views Dialog Box

The **Write Views** dialog box allows you to save selected views to a view file. To open it, click **Write...** in the *Views Dialog Box* (p. 2000). This feature allows you to transfer views between case files. See *Saving Views* (p. 1440) for details.



Controls

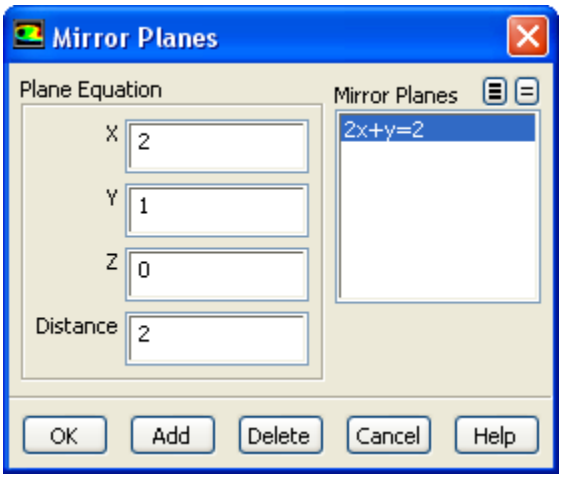
Views to Write

is a selectable list of the defined views. The selected views will be saved to a view file when you click on **OK**.

35.19.26. Mirror Planes Dialog Box

The **Mirror Planes** dialog box allows you to define a symmetry plane for a non-symmetric domain for use with graphics. To open it, click **Define Plane...** in the *Views Dialog Box* (p. 2000). See *Mirroring for Graphics* (p. 1444) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Plane Equation
contains inputs for specifying the equation for the mirror plane: A **X** + B **Y**+ C **Z** = **Distance**.

Mirror Planes
contains a list of all mirror planes you have defined using this dialog box. (Mirror planes that exist in the domain due to symmetry will not appear in this list, since they cannot be modified.)

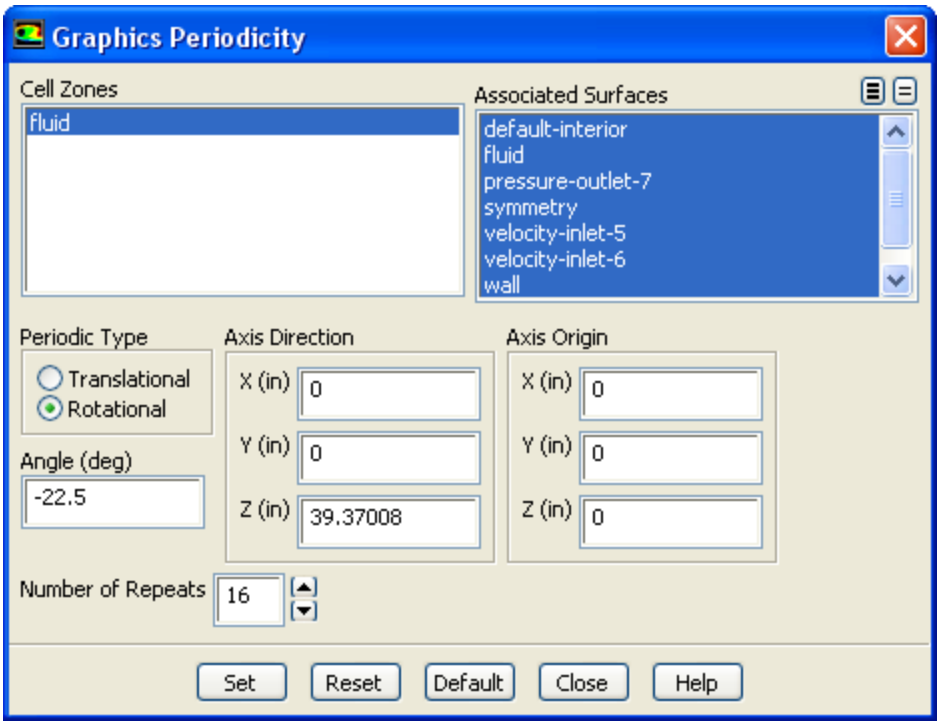
Add
adds the plane defined by the **Plane Equation** to the **Mirror Planes** list.

Delete
deletes the plane(s) selected in the **Mirror Planes** list.

35.19.27. Graphics Periodicity Dialog Box

The **Graphics Periodicity** dialog box allows you to define a periodic repeats, periodic rotation or translation for a non-periodic domain for use with graphics. To open it, click **Define...** under **Periodic Repeats** in the *Views Dialog Box* (p. 2000). See *Periodic Repeats for Graphics* (p. 1443) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Cell Zones

contains the list of the zones in the mesh. You can select one or more zones in this list and specify different periodicity parameters for each zone separately.

Associated Surfaces

contains the list of the surfaces associated with the selected cell zone.

Periodic Type

specifies **Rotational**or **Translational** periodicity.

Angle

specifies the angle by which the domain is rotated to create the periodic repeat. This item is available when you select **Rotational** as the **Periodic Type**.

Translation

specifies the distance in the **X**, **Y**, and **Z** directions by which the domain is translated to create the periodic repeat. This item will appear when you select **Translational** as the **Periodic Type**.

Axis Direction

specifies the direction vector (**X,Y,Z**) for the axis of rotation. This item is available when you select **Rotational** as the **Periodic Type** and the domain is three-dimensional.

Axis Origin

specifies the origin of the axis of rotation. This item is available when you select **Rotational** as the **Periodic Type**.

Number of Repeats

specifies the number of times you want to repeat the periodic domain.

Set

applies the periodicity you have defined for the case setup.

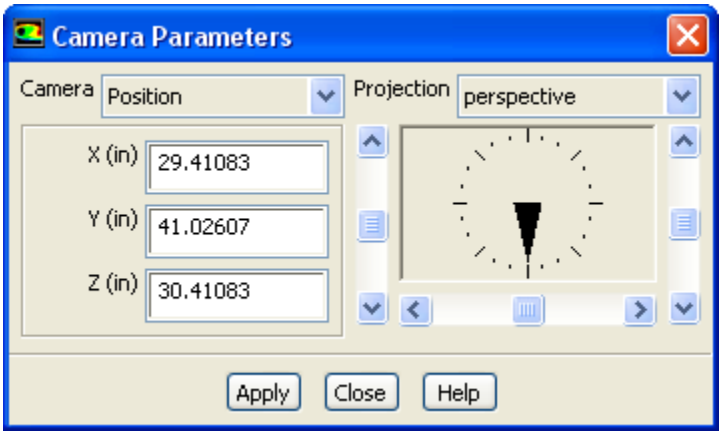
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset

removes any periodicity you have defined, and returns to the default periodicity for the domain (i.e., no periodicity for a non-periodic domain).

35.19.28. Camera Parameters Dialog Box

The **Camera Parameters** dialog box allows you to modify the "camera" through which you are viewing the graphics display. See *Controlling Perspective and Camera Parameters* (p. 1438) for details about the items below.



Controls

Camera

contains a drop-down list of the parameters that define the camera (**Position**, **Target**, **Up Vector**, and **Field**) and **X**, **Y**, and **Z** fields in which you can define the coordinates or field distances for the parameter selected in the drop-down list. *Figure 31.49* (p. 1439) illustrates the definition of the camera by these parameters.

Projection

contains a drop-down list that allows you to select a **Perspective** or **Orthographic** view.

(Dial and Sliders)

allow you to rotate and scale the graphics display. The slider on the scale to the left of the dial rotates the display about the horizontal axis at the center of the screen, the slider on the scale below the dial rotates the display about the vertical axis at the center of the screen, and the dial controls rotation about the axis at the center of and perpendicular to the screen. The slider on the scale to the right of the dial zooms in or out in the display. See *Rotating the Display* (p. 1435) and *Zooming the Display* (p. 1437) for details.

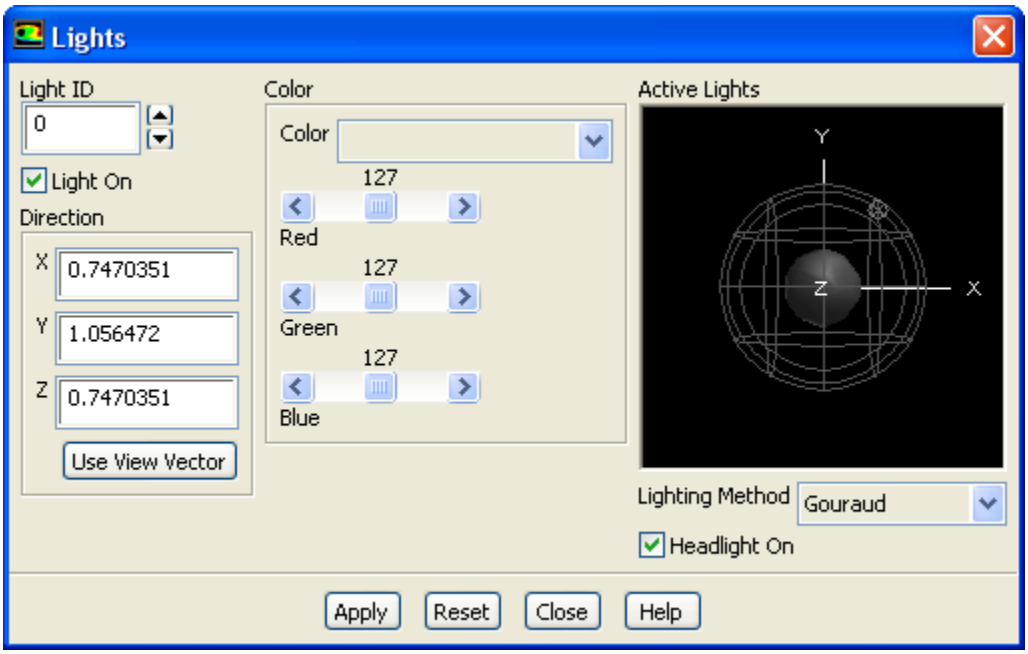
Important

When you are using the sliders and dial to manipulate the view, you may want to turn off **Wireframe Animation** in the *Display Options Dialog Box* (p. 1992), so that you can watch the display move interactively while you are moving the slider or the dial indicator.

35.19.29. Lights Dialog Box

The **Lights** dialog box provides an interactive mechanism for placing colored, directional lights in a scene. See *Adding Lights* (p. 1423) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Light ID

indicates the light that is being added, deleted, or modified. By default, light 1 is defined to be dark gray with a direction of (1,1,1).

Light On

indicates whether or not the light specified in **Light ID**is on or off. By turning off the **Light On** option for a particular light, you can remove this light from the display, while still retaining its definition. To add it to the display again, simply turn on the **Light On** button.

Direction

allows you to specify the direction of the light (the position on the unit sphere from which the light emanates) by entering the **X**, **Y**, and **Z** coordinates or by computing the coordinates based on the current view in the graphics window.

X,Y,Z

specify the direction of the light. For example, the direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin.

Use View Vector

updates the **X,Y,Z** fields with the appropriate values for the current view in the active graphics window, and shines a light in that direction. Instead of entering the **X,Y,Z** values for a light's direction vector, you can use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. You can then click on the **Use View Vector** button to update the **X,Y,Z** fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

Color

allows you to specify the light color with sliders. You can create your desired color by increasing and decreasing the slider values for the colors **Red**, **Green**, and **Blue**. You can also enter a descriptive string (e.g., lavender) in the **Color** field.

Active Lights

shows the position and color of all defined directional lights, and allows you to change the position of a light. All directional lights in ANSYS FLUENT are assumed to be at infinity and pass through the unit

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sphere at the position shown. All light rays arriving at the scene from one light are parallel. The colored markers on the surface of the sphere represent the color and direction of these *distant* lights. These lights point towards the center of the sphere (the origin, which is usually where the geometry is).

Lighting Method

specifies the method to be used in lighting interpolation: **Off**, **Flat**, **Gouraud**, or **Phong**. (**Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.) When **Off** is selected, lighting effects are disabled.

Headlight On

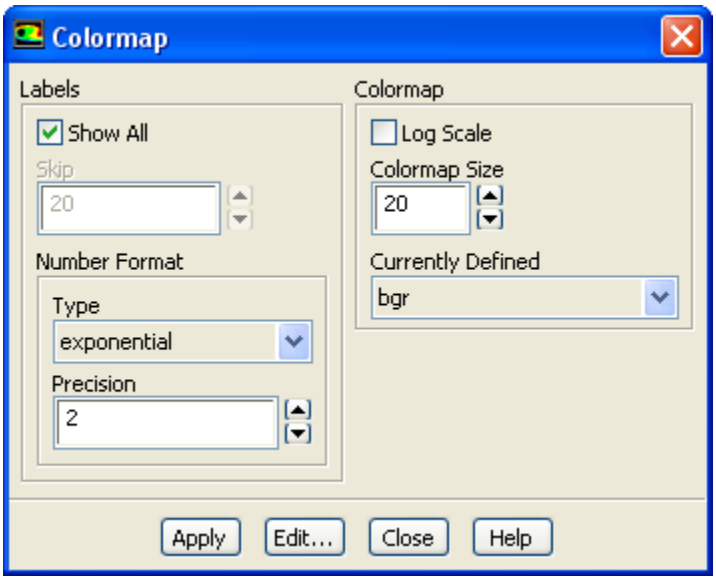
enables constant lighting effects in the direction of the view.

Reset

resets the light definitions to their last saved state (i.e., the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

35.19.30. Colormap Dialog Box

The **Colormap** dialog box allows you to select and modify existing colormaps. See *Selecting a Colormap* (p. 1419) for details.



Controls

Labels

allows you to customize the display of your colormap labels.

Show All

Enable this option if you want all the labels to show alongside the colormap. Disable if you want to skip some of the label displays.

Skip

sets the number of labels to be skipped.

Number Format

contains controls for changing the format of the labels on the color scale. These labels are the character strings used to define the color divisions at the left of the graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type
sets the form of the labels. You may select from a drop-down list of options, including the following:

general
displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float
displays the real value with an integral and fractional part (e.g., 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential
displays the real value with a mantissa and exponent (e.g., 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision
defines the number of fractional digits displayed in the labels.

Colormap
contains controls for the colormap size and scale, and for selection of a defined colormap.

Log Scale
enables the use of a logarithmic scale for the color scale (rather than the default decimal scale). See *Specifying the Colormap Size and Scale* (p. 1419) for details.

Colormap Size
specifies the number of distinct colors in the colormap. You may specify from 2 to 100 colors.

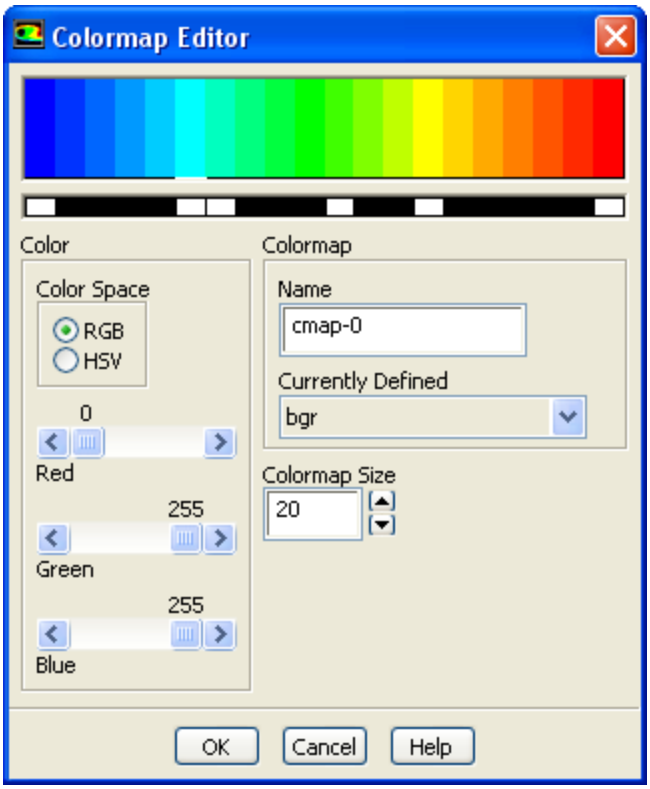
Currently Defined
contains a drop-down list of all pre-defined colormaps and all custom colormaps defined by you. Select the colormap to be used from this list.

Edit...
opens the *Colormap Editor Dialog Box* (p. 2008), in which you can create a custom colormap.

35.19.31. Colormap Editor Dialog Box

The **Colormap Editor** dialog box allows you to create custom colormaps. See *Creating a Customized Colormap* (p. 1421) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

(The Drawing Area)
is used to interactively modify colormaps. You can use the notion of anchor points to interpolate linearly between two defined anchor points. The colors at the top of the dialog box allow you to preview the colormap that is being defined, The black bar and white squares below the colors allow you to set, delete, and modify anchor points.

Color
specifies color components for the currently selected anchor point.

Color Space
gives you a choice of selecting the color specification.

RGB
enables the specification of colors based on their red, green, and blue components.

HSV
enables the specification of colors based on their hue, saturation, and value.

Red, Green, Blue
are scales with which you can specify the RGB components of the selected anchor color. These scales will appear when **RGB** is selected.

Hue, Saturation, Value
are scales with which you can specify the HSV components of the selected anchor color. These scales will appear when **HSV** is selected.

Colormap
lists colormap names and the name of the colormap being edited.

Name
sets the name of the colormap being edited/selected. You can rename existing colormaps and provide names for new colormaps.

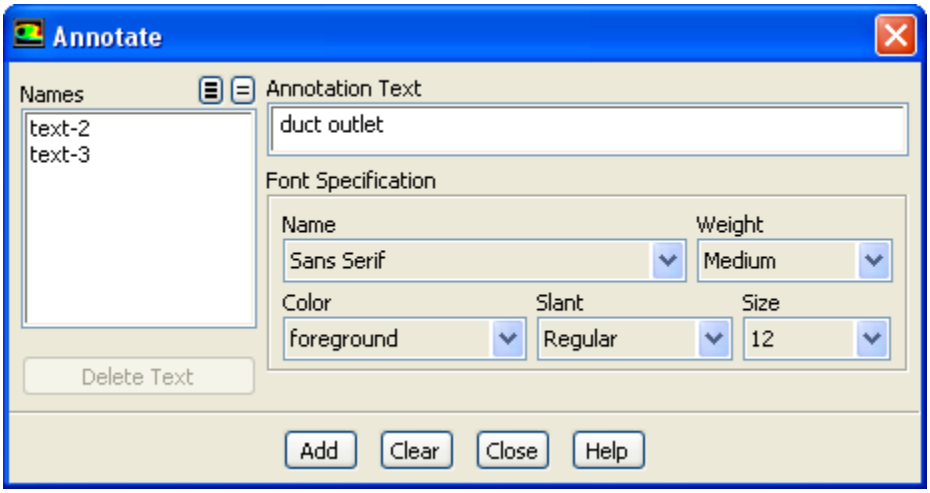
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Currently Defined
contains a drop-down list of all pre-defined colormaps and all custom colormaps defined by you. You can select a colormap to be modified from this list.

Colormap Size
specifies the number of distinct colors in the colormap. You may specify from 2 to 100 colors.

35.19.32. Annotate Dialog Box

You can use the **Annotate** dialog box to add text with optional attachment lines to the graphics windows, or to modify existing text. See *Adding Text Using the Annotate Dialog Box* (p. 1415) for details about the items below.



Controls

Names
contains a selectable list of all annotation text strings that have been defined. You can choose a text string to be deleted or edited.

Delete Text
deletes the text strings selected in the **Names** list from the display.

Annotation Text
contains the annotation text string you wish to add, or the annotation text string for the item selected in the **Names** list.

Font Specification
contains controls for defining or modifying the font in the annotation text string.

Name
contains a drop-down list of various font styles.

Weight
contains a drop-down list from which you can select **Medium** or **Bold**.

Color
contains a drop-down list of colors that can be used for the text.

Slant
contains a drop-down list from which you can select **Regular** or **Italic** as the slant type.

Size
contains a drop-down list of font sizes (in points).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add

adds the current **Annotation Text** to the active graphics window. A dialog box will prompt you to select a screen location using the mouse-probe button on your mouse (see *Controlling the Mouse Button Functions* (p. 1427) for more information on setting the mouse buttons).

Edit

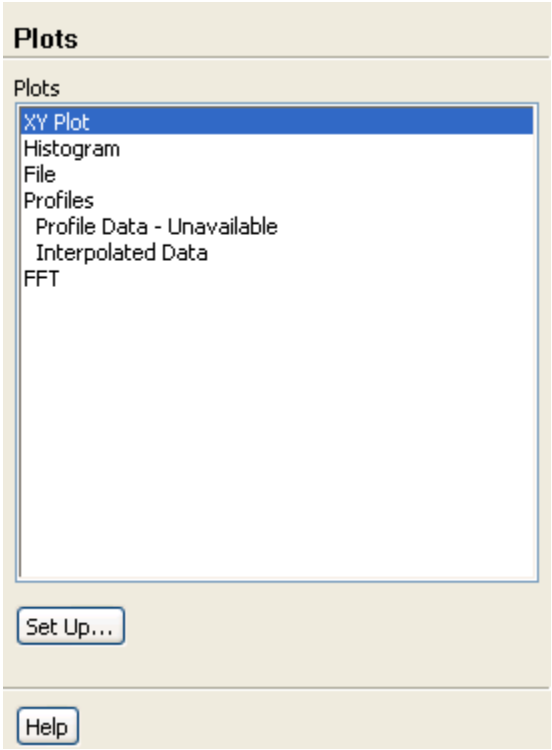
updates the edited text in the active graphics window. This button will replace the **Add** button when you are editing an existing text string from the **Names** list.

Clear

removes all annotation text and attachment lines from the active graphics window.

35.20. Plots Task Page

The **Plots** task page allows you to create plots (XY, histograms, profiles, etc.) of your computational results. See *Displaying Graphics* (p. 1379) for more information.



Controls

Plots

contains a listing of the various plot types available in ANSYS FLUENT.

You can double-click an item in the **Plots** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

XY Plot

selecting this item and clicking the **Set Up...** button opens the *Solution XY Plot Dialog Box* (p. 2012).

Histogram

selecting this item and clicking the **Set Up...** button opens the *Histogram Dialog Box* (p. 2015).

File

selecting this item and clicking the **Set Up...** button opens the *File XY Plot Dialog Box* (p. 2016).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Profiles

two types of profile plots are available:

Profile Data

selecting this item and clicking the **Set Up...** button opens the *Plot Profile Data Dialog Box* (p. 2017).

Interpolated Data

selecting this item and clicking the **Set Up...** button opens the *Plot Interpolated Data Dialog Box* (p. 2018).

FFT

selecting this item and clicking the **Set Up...** button opens the *Fourier Transform Dialog Box* (p. 2019).

Set Up...

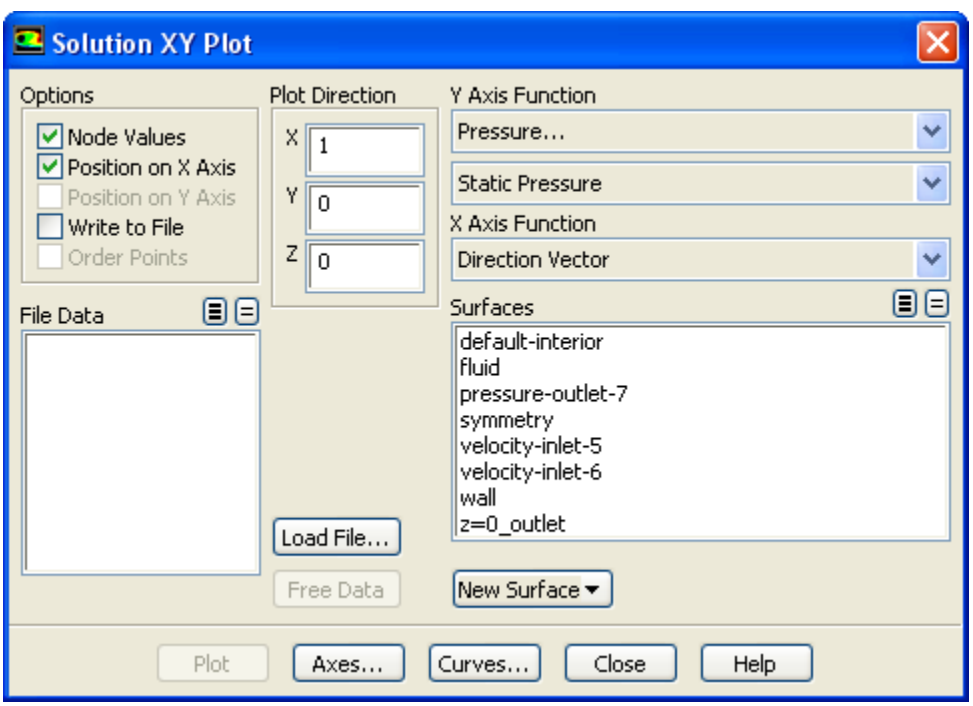
displays the dialog box corresponding to the selected item in the **Plots** list.

For additional information, please see the following sections:

35.20.1. Solution XY Plot Dialog Box
35.20.2. Histogram Dialog Box
35.20.3. File XY Plot Dialog Box
35.20.4. Plot Profile Data Dialog Box
35.20.5. Plot Interpolated Data Dialog Box
35.20.6. Fourier Transform Dialog Box
35.20.7. Plot/Modify Input Signal Dialog Box
35.20.8. Axes Dialog Box
35.20.9. Curves Dialog Box

35.20.1. Solution XY Plot Dialog Box

The **Solution XY Plot** dialog box allows you to display zone, surface and file data in an XY plot format. See *Steps for Generating Solution XY Plots* (p. 1468) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

contains check buttons that control the presentation of node or cell-averaged values, the selection of axis functions, and the ability to write the plot data to a file.

Node Values

toggles the node averaging of the data presented in the plot. If the option is inactive, cell values are presented. See *Choosing Node or Cell Values* (p. 1472) for details.

Position on X Axis, Position on Y Axis

set the x-axis or y-axis function to be the position. If one of these options is turned on, the other will automatically be turned off. You can turn both options off to generate a plot of one flow-field function vs. another, selecting the function for each axis using the **X Axis Function** and **Y Axis Function** drop-down lists.

Write to File

activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open the **Select File** dialog box (*The **Select File** Dialog Box* (p. 33) and *The **Select File** Dialog Box (Windows)* (p. 33)), in which you can specify a name and save a file containing the plot data. The format of this file is described in *XY Plot File Format* (p. 1477).

Order Points

specifies that plot data being saved to a file should be sorted in order of ascending x axis value. This option is available only when the **Write to File** option is turned on.

Plot Direction

contains inputs for defining the plot direction.

If **Direction Vector** (the default) is selected in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), the inputs are the components of the direction vector. The position axis of the plot will have coordinate values that correspond to the dot product of the data coordinate vector with the plot direction vector. See *Steps for Generating Solution XY Plots* (p. 1468) for details.

X

is the component in the x direction.

Y

is the component in the y direction.

Z

is the component in the z direction.

If **Curve Length** is selected in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), the inputs are the direction along the length of the surface selected in the **Surfaces** list. See *Steps for Generating Solution XY Plots* (p. 1468) for details.

Default

specifies the plot direction as the direction of increasing curve length.

Reverse

specifies the plot direction as the direction of decreasing curve length.

Show

displays the selected surface in the graphics window, marking the start of the surface with a blue dot and the end of the surface with a red dot. ANSYS FLUENT will also display arrows on the surface showing the direction in which the variable will be plotted.

Y Axis Function, X Axis Function

contain lists of solution variables that can be used for the y or x axis of the plot. If the **Position on X Axis** option is turned on, **X Axis Function** will become a single drop-down list, containing two options:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Vector (to plot the selected variable as a function of position along a specified direction vector) and **Curve Length** (to plot the selected variable as a function of position along the length of a specified curvilinear surface). See *Steps for Generating Solution XY Plots* (p. 1468) for details.

Likewise, if **Position on Y Axis** is turned on, **Y Axis Function** will become a single drop-down list containing **Direction Vector** and **Curve Length**. If both **Position on X Axis** and **Position on Y Axis** are turned off, you can select field functions for both axes using the **X Axis Function** and **Y Axis Function** lists.

File Data
is a selectable list of the plot titles associated with the loaded external data files. You may choose any number of files for the data plot. The files are loaded using the **Load File...** push button. The format of these files is presented in *XY Plot File Format* (p. 1477). See *Including External Data in the Solution XY Plot* (p. 1472) for details.

Load File...
opens the **Select File** dialog box (*The **Select File** Dialog Box* (p. 33)), in which you can select the plot file to be read. See *Including External Data in the Solution XY Plot* (p. 1472) for details. After the external file is loaded, its plot title will be displayed in the **File Data** list.

Free Data
removes the files selected in the **File Data** list.

Surfaces
is a selectable list of surfaces in the solution domain. You may choose any number of surfaces for the data plot.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the *Point Surface Dialog Box* (p. 1927).

Line/Rake
opens the *Line/Rake Surface Dialog Box* (p. 1928).

Plane
opens the *Plane Surface Dialog Box* (p. 1929).

Quadric
opens the *Quadric Surface Dialog Box* (p. 1931).

Iso-Surface
opens the *Iso-Surface Dialog Box* (p. 1933).

Iso-Clip
opens the *Iso-Clip Dialog Box* (p. 1935).

Plot
plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes. If the **Write to File** option is turned on, this button becomes the **Write...** button.

Write...
opens the **Select File** dialog box (*The **Select File** Dialog Box* (p. 33)), in which you can save the plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

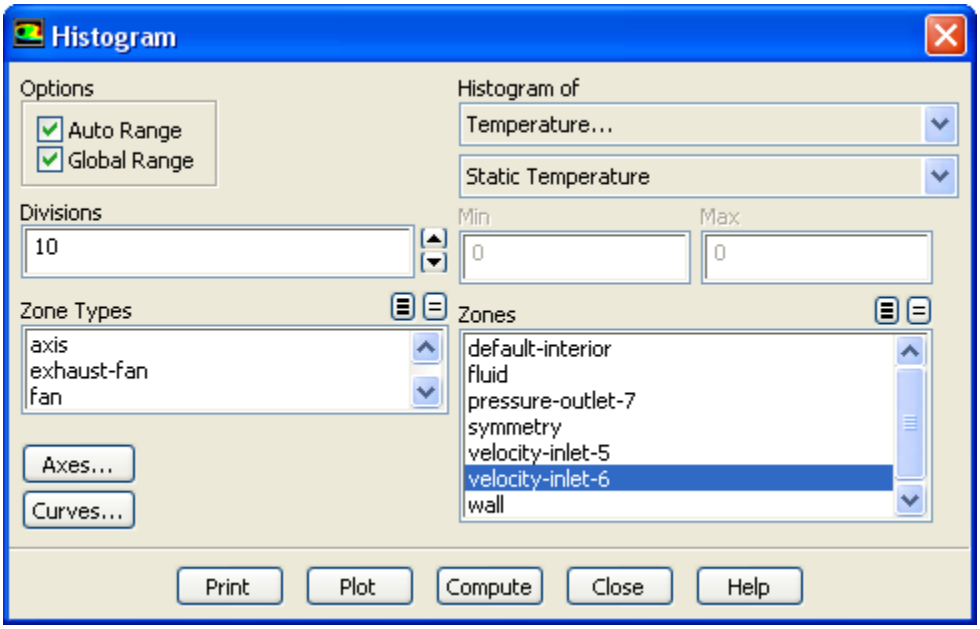
Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.20.2. Histogram Dialog Box

The **Histogram** dialog box allows you to create histograms of selected geometric or physical data. See *Steps for Generating Histogram Plots* (p. 1478) for details about the items below.



Controls

Options

contains the check buttons for current histogram options.

Auto Range

toggles the ability to specify the minimum and maximum range of scalar values in the histogram print or plot. If the option is not active, the **Min** and **Max** fields are editable, and you may specify the desired range. If the option is active, the range is defined by the minimum and maximum values in the computational domain.

Global Range

toggles the ability to specify the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain.

Divisions

sets the number of data intervals that will exist in the histogram.

Zone Types

contains a list of available face or cell zone types.

Axes...

opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Curves...

opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the histogram plot.

Histogram of

contains a list of scalar quantities that can be used in the histogram.

Min

displays or allows definition of the minimum value of the selected scalar quantity used in the histogram.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max
displays or allows definition of the maximum value of the selected scalar quantity used in the histogram.

Zones
contains a list of available face or cell zones.

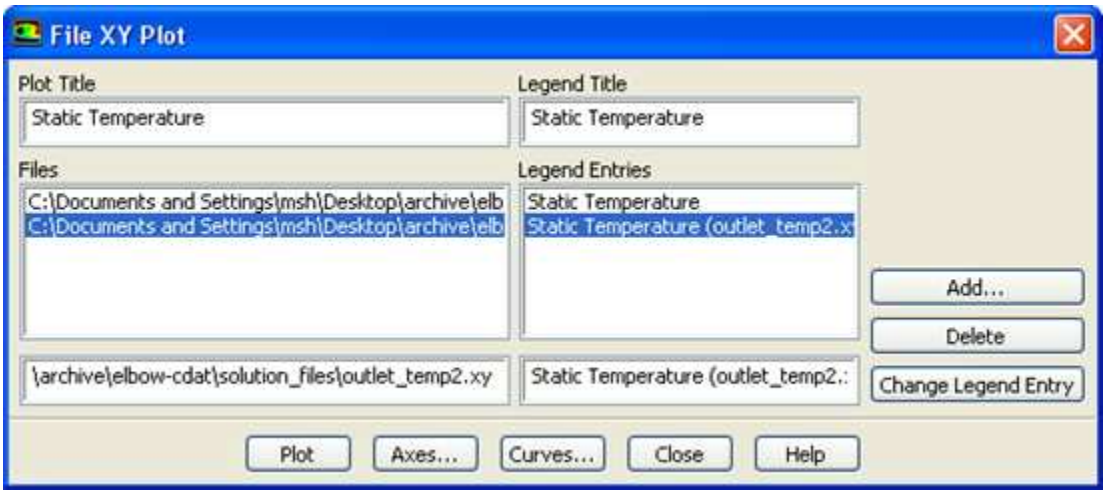
Print
displays the histogram interval.

Plot
displays a plot of the percentage of the total number of cells versus the scalar quantity in the active graphics window.

Compute
computes the minimum and maximum cell values of the selected scalar quantity. The values are displayed in **Min** and **Max**.

35.20.3. File XY Plot Dialog Box

The **File XY Plot** dialog box allows you to display data read from one or more files in an abscissa/ordinate plot form. The format of the plot file is described in *XY Plot File Format* (p. 1477). See *Steps for Generating XY Plots of Data in External Files* (p. 1472) for details about the items below.



Controls

Plot Title
displays the current plot title. The default plot title is the y-axis label of the first file read into the dialog box, but you can edit the text entry to change the plot title.

Files
contains a selectable list of loaded file names. If you enter a valid file name into the text entry field below the list and press the **Add...** button ANSYS FLUENT will load the file and update the dialog box. If, however, the file is not found or has already been read, the *The **Select File** Dialog Box* (p. 33) will be opened.

Legend Title
sets the title of the legend. By default, the legend has no title.

Legend Entries
contains a selectable list of legend labels associated with the loaded files. The default legend label is the title (first string) in the file. You can modify the legend label by selecting the old name in the **Legend Entries** list. It is then displayed in the text entry field below the list and may be edited.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add...
loads the file named in the **Files** text entry field or opens the **Select File** dialog box if no name, a wrong name, or a duplicate name appears in that text field.

Delete
removes the file selected in the **Files** list.

Change Legend Entry
changes the legend label of the selected file to the text entered in the text entry field below the **Legend Entries** list.

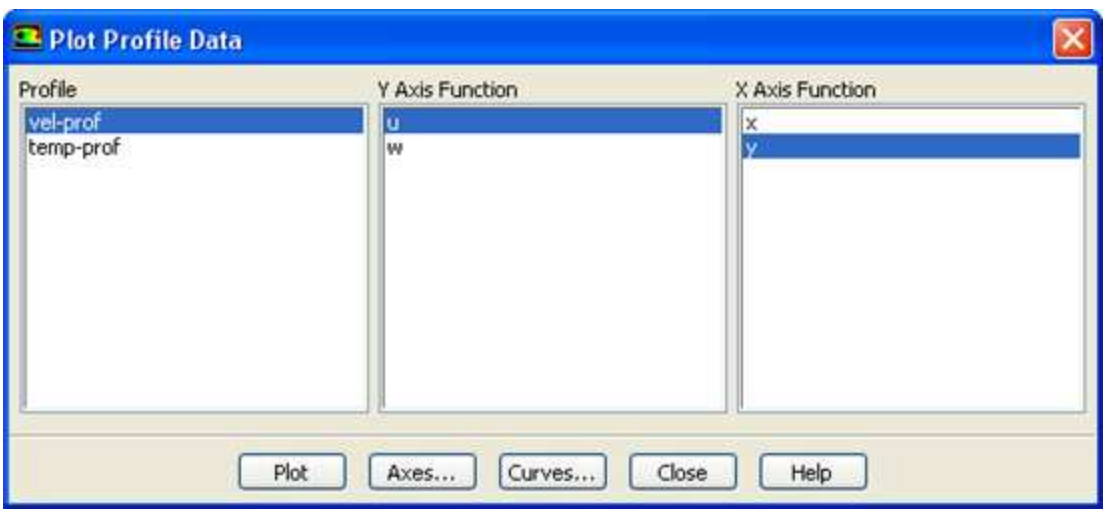
Plot
displays an XY plot of the data associated with every loaded file. You can use the **Delete** button to remove files that you do not wish to plot.

Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot.

35.20.4. Plot Profile Data Dialog Box

The **Plot Profile Data** dialog box allows you to display an XY plot of the original data points of a boundary profile before it is interpolated onto the cell faces of a boundary. See *Steps for Generating Plots of Profile Data* (p. 1474) for details about the items below.



Controls

Profile
contains a selectable list of available profiles. When a profile is selected its available fields are displayed under **Y Axis Function**.

Y Axis Function
contains a selectable list of the fields available in the selected profile that can be used for the y axis of the plot.

X Axis Function
contains a selectable list of the variables that can be used for the x axis of the plot.

Plot
displays an XY plot of the data points from the selected profile.

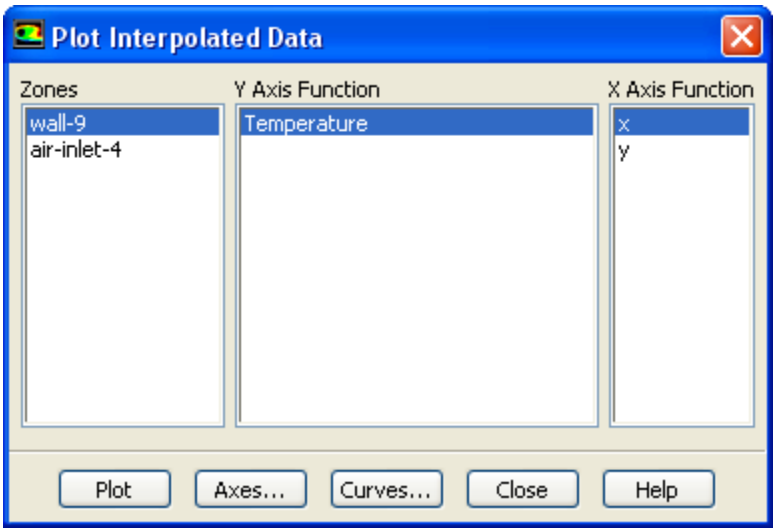
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot.

35.20.5. Plot Interpolated Data Dialog Box

The **Plot Interpolated Data** dialog box allows you to display an XY plot of the values assigned to the cell faces when a profile file has been interpolated on a boundary. See *Steps for Generating Plots of Interpolated Profile Data* (p. 1475) for details about the items below.



Controls

Zones
contains a selectable list of the zones for which a profile field has been set as one or more of the parameters are displayed under **Y Axis Function**.

Y Axis Function
contains a selectable list of the profile-related parameters in the selected zone that can be used for the y axis of the plot. The name of the parameter will be the same as that of the drop-down list in the boundary condition dialog box from which the profile field was selected.

X Axis Function
contains a selectable list of the geometry variables that can be used for the x axis of the plot.

Plot
displays an XY plot of the cell face values (as interpolated from the data points of the profile file) of the selected zone.

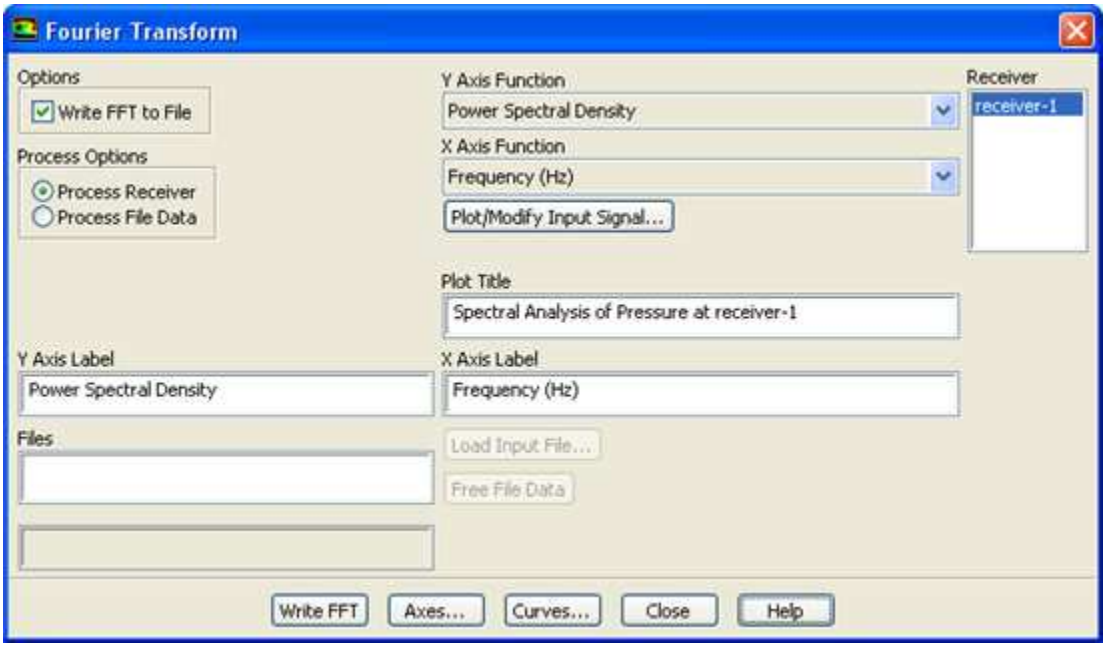
Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.20.6. Fourier Transform Dialog Box

The **Fourier Transform** dialog box allows you to analyze your time dependent data using the Fast Fourier Transform (FFT) algorithm. See *Fast Fourier Transform (FFT) Postprocessing* (p. 1500) for details about the items below.



Controls

Options

lets you write the FFT data to a file or display the FFT data in a graphics window.

Write FFT to File

allows you to write out the FFT data directly to a file. When selected, the **Plot FFT...** buttons becomes the **Write FFT...** button.

Process Options

contains options to analyze signal data.

Process Receiver

allows you to analyze receiver data stored in memory.

Process File Data

allows you to analyze signal data from an existing input file.

Y Axis Function

contains a list from which you can select the function for the y axis.

X Axis Function

contains a list from which you can select the function for the x axis.

Receiver

contains a list of receivers from which you can select when the **Process Receiver** option is enabled.

Plot/Modify Input Signal...

opens the *Plot/Modify Input Signal Dialog Box* (p. 2020), in which you can set additional options for vector displays.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Title
lets you create a new title or edit the original title for the FFT plot. By default, ANSYS FLUENT adds the string "Spectral Analysis of" to the title originally applied to the input signal plot.

Y-axis Label
allows you to create a new y axis label or edit the original y axis label. By default, the **Y-axis Label** corresponds to the selection in the **Y Axis Function** drop-down list.

X-axis Label
allows you to create a new x axis label or edit the original x axis label. By default, the **X-axis Label** corresponds to the selection in the **X Axis Function** drop-down list.

Files
lists the loaded input signal data files.

Load Input File...
loads an input signal data file into ANSYS FLUENT for FFT analysis. The input file is listed under **Files**.

Free File Data
removes data from FFT analysis once the input signal data file is selected in the **Files** list.

Plot FFT
displays the FFT data in a graphic window. If the **Write FFT to File** option is selected, then this button becomes the **Write FFT** button, which opens a file selection dialog box so that you write the FFT data to a file.

Write FFT
opens *The **Select File** Dialog Box* (p. 33) in which you can specify a name and save the FFT data to a file. If the **Write FFT to File** option is selected, then the **Plot FFT** button changes to the **Write FFT** button.

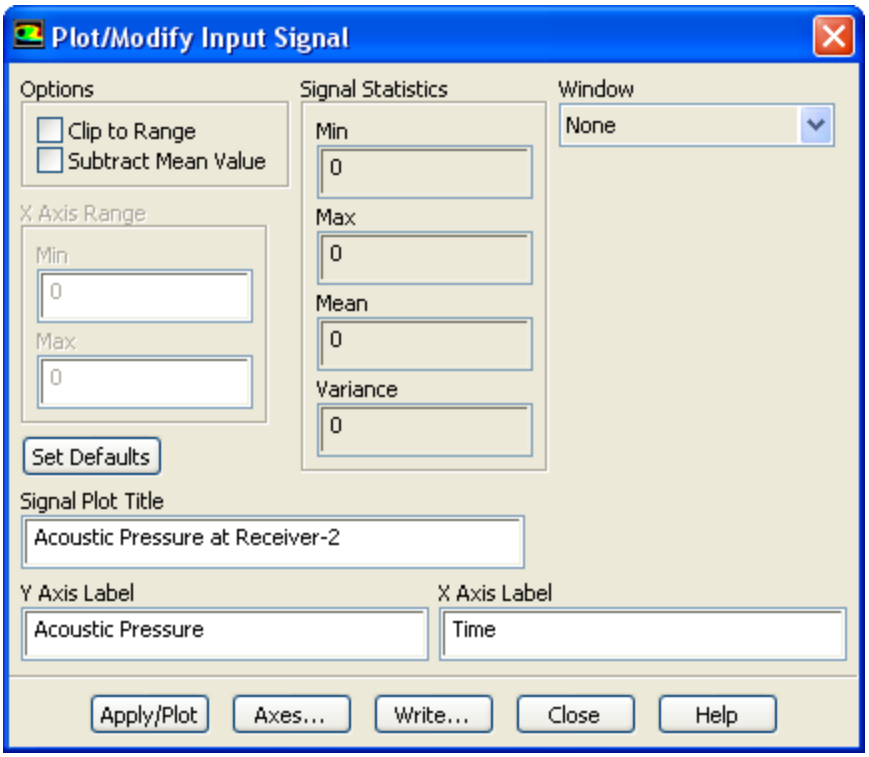
Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the plot.

35.20.7. Plot/Modify Input Signal Dialog Box

The **Plot/Modify Input Signal** dialog box allows you to customize the input signal data set. It is opened from the *Fourier Transform Dialog Box* (p. 2019). See *Using the FFT Utility* (p. 1503) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

allows you to process some or all of the input signal data.

Clip to Range

allows you to analyze a portion of the input signal by specifying data range.

Subtract Mean Value

reduces y axis quantities by the mean value of the relevant signal property.

Signal Statistics

displays signal information.

Min

displays the minimum value for the input signal.

Max

displays the maximum value for the input signal.

Mean

displays the average value for the input signal.

Variance

displays the variance for the input signal.

X Axis Range

allows for a portion of the input signal to be analyzed when the **Process the whole data** option is turned off.

Min

specifies a minimum value for the x axis.

Max

specifies a maximum value for the x axis.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set Defaults
resets the x axis minimum and maximum values to their original value and turns on the **Process the whole data** option.

Signal Plot Title
lets you create a new title or edit the original title for the input signal plot. By default, the **Signal Plot Title** corresponds with the title originally applied to the input signal plot.

Y Axis Label
allows you to create a new y axis label or edit the original y axis label. By default, the **Y-axis Label** corresponds with the y axis label originally applied to the input signal plot.

X Axis Label
allows you to create a new x axis label or edit the original x axis label. By default, the **X-axis Label** corresponds with the x axis label originally applied to the input signal plot.

Window
lets you specify a windowing technique to remove discontinuities in the FFT calculation.

None
applies no windowing technique. This is the default setting.

Hamming
applies the Hamming technique.

Hanning
applies the Hanning technique.

Barlett
applies the Barlett technique.

Blackman
applies the Blackman technique.

Apply/Plot
applies any changes you have made in the dialog box and displays the input signal data in a graphics window.

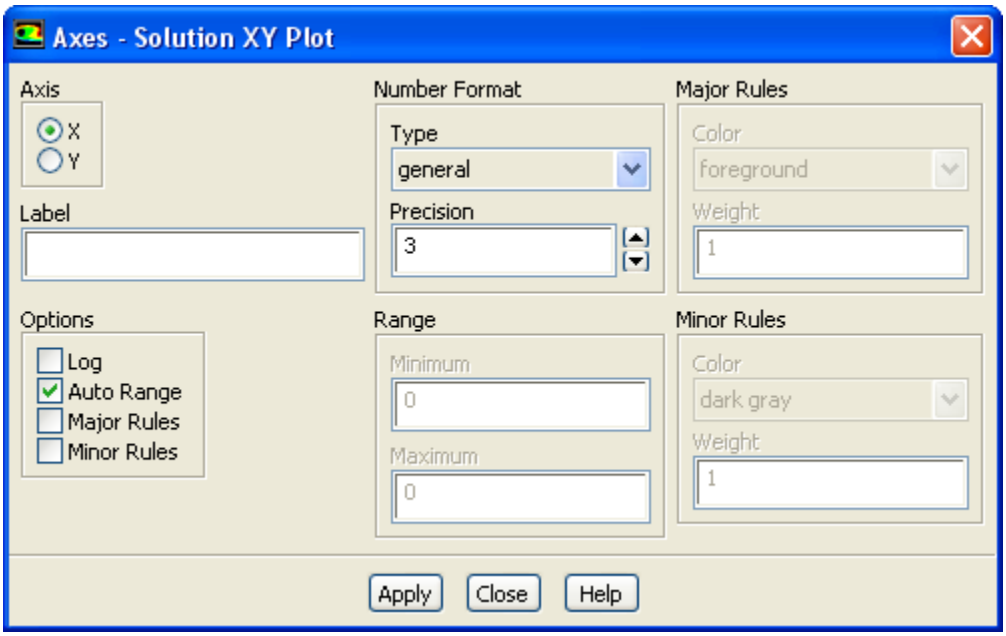
Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

Write...
opens *The **Select File** Dialog Box* (p. 33), in which you can save the signals to a file.

35.20.8. Axes Dialog Box

The **Axes** dialog box allows you to independently control the characteristics of the ordinate and abscissa on an XY plot or histogram. You can change the labels, scale, range, number format, and major and minor rules visibility and appearance. Note that the title following **Axes** in the dialog box indicates which plot environment you are changing. You can set different parameters for each type of plot that ANSYS FLUENT can generate. See *Using the Axes Dialog Box* (p. 1480) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Axis

contains check buttons that allow you to set abscissa (x-axis) or ordinate (y-axis) characteristics.

X

allows you to specify the abscissa characteristics.

Y

allows you to specify the ordinate characteristics.

Label

defines the character string that will label the active axis (the one selected in **Axis**) in the display.

Options

contains check buttons that (de)activate scale, range, major rules, and minor rules.

Log

toggles logarithmic scaling of the active axis. By default, decimal scaling is used.

Auto Range

toggles automatic computation of the range of the active axis. If you deactivate this option, you may input the **Minimum** and **Maximum** values in the **Range** box.

Major Rules

toggles the display of major rules on the active axis. Major rules are the horizontal or vertical lines that mark the primary data divisions and span the whole plot window to produce a "grid."

Minor Rules

toggles the display of minor rules on the active axis. Minor rules are the horizontal or vertical lines that mark the secondary data divisions and span the whole plot window to produce a "grid."

Number Format

contains controls for changing the format of the data labels on the active axis. Data labels are the character strings used to define the primary data divisions on the axes.

Type

sets the form of the data labels. You may select from a drop-down list of options, including the following:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

general
displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float
displays the real value with an integral and fractional part (e.g., 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential
displays the real value with a mantissa and exponent (e.g., 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision
defines the number of fractional digits displayed in the data labels.

Range
contains the range or extents of the active axis. To set the range manually, you must turn off **Auto Range**. Otherwise the extents are computed automatically.

Minimum
sets the minimum data value for the active axis.

Maximum
sets the maximum data value for the active axis.

Major Rules
contains controls for modifying the appearance of the major rules. To use these controls you must activate **Major Rules** in the **Options** list.

Color
sets the color of the major rules from a drop-down list with numerous color selections.

Weight
sets the line thickness of the major rule. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

Minor Rules
contains controls for modifying the appearance of the minor rules. To use these controls you must activate **Minor Rules** in the **Options** list.

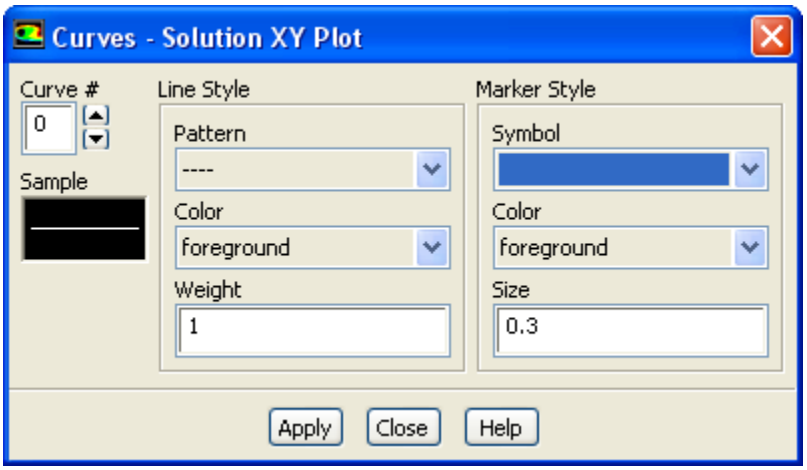
Color
sets the color of the minor rules from a drop-down list with numerous color selections.

Weight
sets the line thickness of the minor rule. A line of weight 1.0 is 1 pixel wide. A weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

35.20.9. Curves Dialog Box

The **Curves** dialog box allows you to modify the appearance of the lines and markers used in XY plots. Note that the title following **Curves** in the dialog box indicates which plot environment you are changing. You can set different parameters for each type of plot that ANSYS FLUENT can generate. See *Using the Curves Dialog Box* (p. 1482) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Curve #

defines the active curve number. The present and future marker and line styles apply to the defined curve number. The curves are numbered sequentially, starting from 0. For example, if you were plotting flow-field values on two zones, the first zone would be curve 0, and the second, curve 1. If the plot contains only one curve, the **Curve #** is set to 0 and is not editable.

Sample

displays a single marker and line with the current style attributes.

Line Style

contains controls for modifying the appearance of the active curve.

Pattern

sets the pattern of the active curve. A drop-down list allows you to set the line pattern. Except for **center** and **phantom** lines, the list displays examples of the pattern choices. A **center**line alternates a very long dash and a short dash and a **phantom** line alternates a very long dash and a double short dash.

Color

sets the color of the active curve. A drop-down list allows you to select from a list of color names.

Weight

sets the thickness of the active curve. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

Marker Style

contains controls for modifying the appearance of the active curve's marker.

Symbol

sets the symbol used to mark data. You can select the symbol from a drop-down list that contains all the symbol choices. The **Sample** box will allow you to experiment with various markers. For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol `/*\` (filled-in upward-pointing triangle) could be used for the marker representing the upper surface data, and the symbol `\*/` (filled-in downward-pointing triangle) could be used for the marker representing the lower surface data.

Color

sets the color of the marker on the active line number. A drop-down list allows you to select from a list of color names.

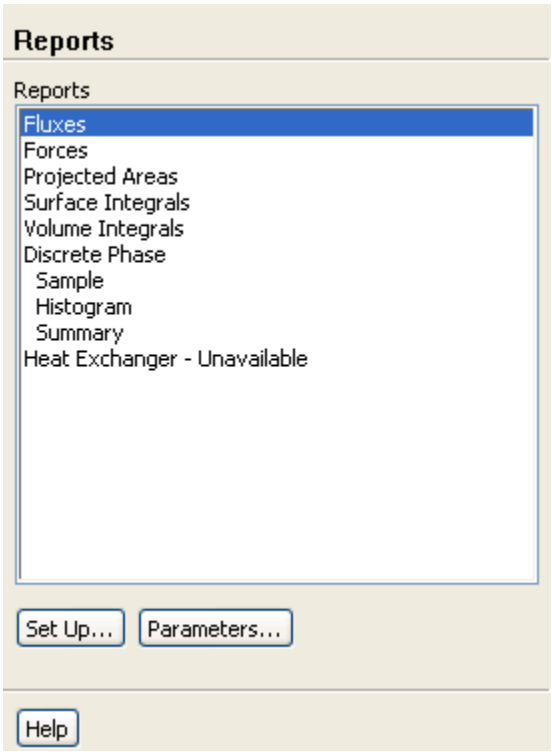
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Size

sets the size of the data marker. A symbol of size 1.0 is 3.0% of the height of the display screen, except for the "." symbol, which is always one pixel.

35.21. Reports Task Page

The **Reports** task page allows you to set up reports for your CFD simulation. Reports can be compiled for fluxes, forces, projected areas, surface and volume integrals, among others. See *Reporting Alphanumeric Data* (p. 1509) for more information.



Controls

Reports

displays a list of available report types in ANSYS FLUENT.

You can double-click an item in the **Reports** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Fluxes

selecting this item and clicking the **Set Up...** button opens the *Flux Reports Dialog Box* (p. 2027).

Forces

selecting this item and clicking the **Set Up...** button opens the *Force Reports Dialog Box* (p. 2029).

Projected Areas

selecting this item and clicking the **Set Up...** button opens the *Projected Surface Areas Dialog Box* (p. 2030).

Surface Integrals

selecting this item and clicking the **Set Up...** button opens the *Surface Integrals Dialog Box* (p. 2031).

Volume Integrals

selecting this item and clicking the **Set Up...** button opens the *Volume Integrals Dialog Box* (p. 2034).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase
allows you to report on one of the following three report types:

Sample
selecting this item and clicking the **Set Up...** button opens the *Sample Trajectories Dialog Box* (p. 2036).

Histogram
selecting this item and clicking the **Set Up...** button opens the *Trajectory Sample Histograms Dialog Box* (p. 2038).

Summary
selecting this item and clicking the **Set Up...** button opens the *Particle Summary Dialog Box* (p. 2039).

Heat Exchanger
selecting this item and clicking the **Set Up...** button opens the *Heat Exchanger Report Dialog Box* (p. 2040).

Set Up...
opens the dialog box corresponding to the selected item in the **Reports** list.

Parameters...
opens the *Parameters Dialog Box* (p. 2041).

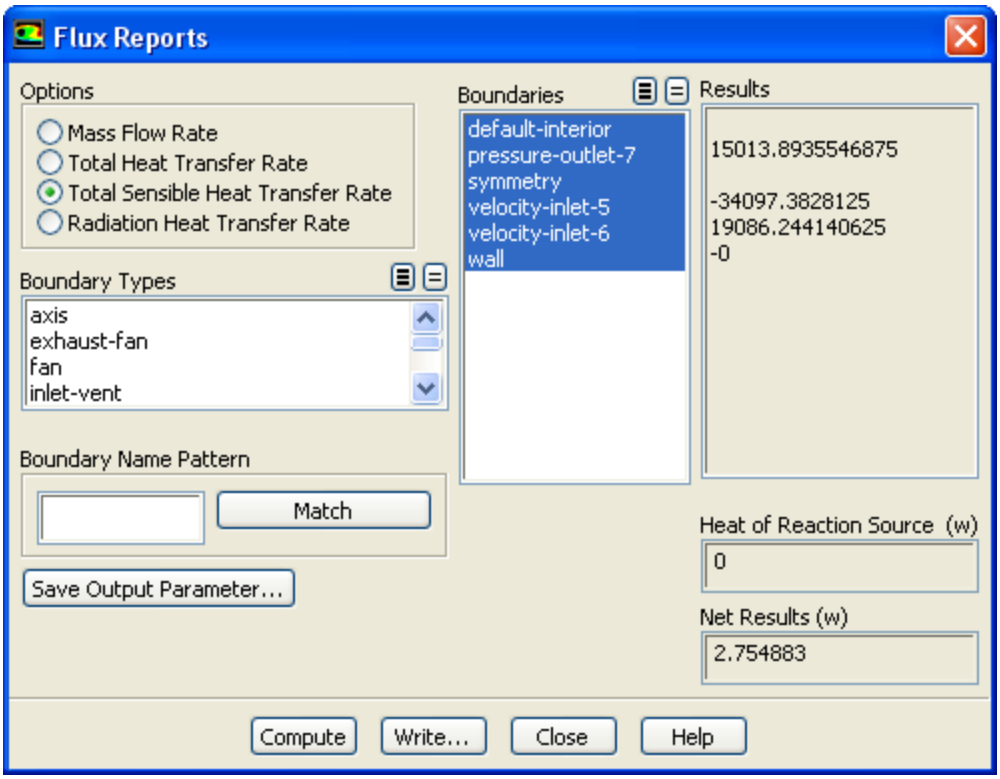
For additional information, please see the following sections:

35.21.1. Flux Reports Dialog Box
35.21.2. Force Reports Dialog Box
35.21.3. Projected Surface Areas Dialog Box
35.21.4. Surface Integrals Dialog Box
35.21.5. Volume Integrals Dialog Box
35.21.6. Sample Trajectories Dialog Box
35.21.7. Trajectory Sample Histograms Dialog Box
35.21.8. Particle Summary Dialog Box
35.21.9. Heat Exchanger Report Dialog Box
35.21.10. Parameters Dialog Box
35.21.11. Rename Dialog Box
35.21.12. Input Parameter Properties Dialog Box
35.21.13. Save Output Parameter Dialog Box

35.21.1. Flux Reports Dialog Box

The **Flux Reports** dialog box allows you to compute the mass flow rate, heat transfer rate, and radiation heat transfer rate on selected boundary zones. See *Fluxes Through Boundaries* (p. 1510) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the following radio buttons:

Mass Flow Rate

turns on the computation of the mass flow rate for the selected boundary zones.

Total Heat Transfer Rate

turns on the computation of the total heat transfer rate for the selected boundary zones.

Total Sensible Heat Transfer Rate

turns on the computation of the total sensible heat transfer rate for the selected boundary zones. It reports the total energy flux as defined in Equation 5–2 in the Theory Guide.

Radiation Heat Transfer Rate

turns on the computation of the radiation heat transfer rate for the selected boundary zones.

Boundary Types

contains a selectable list of types of boundary zones. If you select (or deselect) an item in this list, all zones of that type will be selected (or deselected) automatically in the **Boundaries** list.

Boundary Name Pattern

specifies the pattern to look for in the names of boundary zones. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Boundaries** list with names that match the specified pattern. See *Generating a Flux Report* (p. 1511) for information about matching additional characters using * and ?.

Save Output Parameter...

opens the *Save Output Parameter Dialog Box* (p. 2044).

Boundaries

contains a selectable list of valid boundary zones for flux reporting.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results

displays the results of the selected flux computation for each boundary zone selected. The summation of the individual zone flux results is displayed in the box below the **Results** list.

Compute

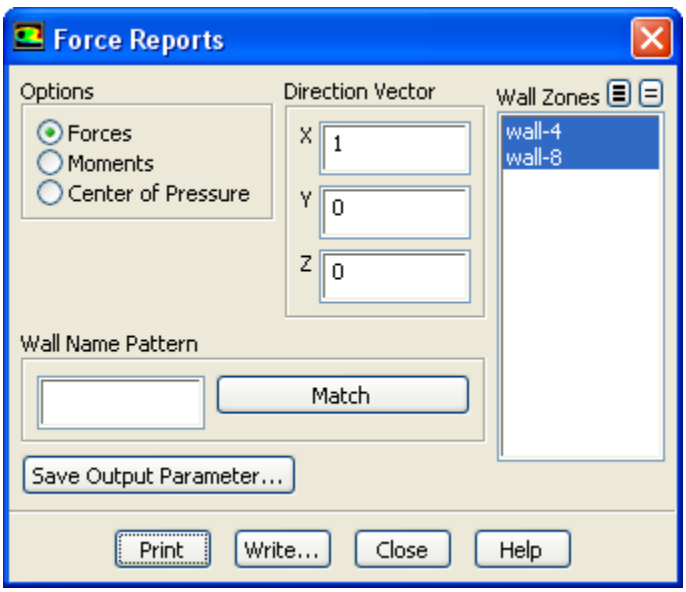
computes the flux for each of the selected boundary zones and updates the **Heat of Reaction Source** and **Net Results** (e.g., in kg/s, W, etc.).

Write...

opens the **Select File** dialog box (*The **Select File** Dialog Box* (p. 33) and *The **Select File** Dialog Box (Windows)* (p. 33)), which you can use to save the reported values to a file.

35.21.2. Force Reports Dialog Box

The **Force Reports** dialog box allows you to compute the forces along a specified vector, moments about a specified center, and the coordinates of the center of pressure for selected wall zones. See *Forces on Boundaries* (p. 1515) for details.



Controls

Options

contains radio buttons that control computation of the forces, moments, or center of pressure.

Forces

enables the computation of the pressure and viscous forces.

Moments

enables the computation of the pressure and viscous moments.

Center of Presure

enables the computation of the average location of the pressure.

Direction Vector

contains the components of the force vector. This label is visible when the **Forces** radio button is active.

X,Y,Z

are the components of the force vector along which the forces will be computed.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moment Center
contains the Cartesian coordinates of the moment center. This label is visible when the **Moments** radio button is active.

X,Y,Z
are the Cartesian coordinates of the moment center about which moments will be computed.

Moment Axis
contains the Cartesian coordinates of the moment axis. This label is visible when the **Moments** radio button is active.

X,Y,Z
are the Cartesian coordinates of the moment axis about which moments will be computed.

Coordinate
contains the value of the Cartesian coordinate that is fixed. This label is visible when the **Center of Pressure** radio button is active.

X,Y,Z
are the Cartesian coordinates, one of which will be fixed.

Value (n)
is the point where the selected Cartesian coordinate will be fixed.

Wall Zones
contains a selectable list of wall zones. The force or moment information is printed for each zone, and then a total force or moment for all the zones is presented.

Wall Name Pattern
specifies the pattern to look for in the names of wall zones. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Wall Zones** list with names that match the specified pattern. See *Generating a Force, Moment, or Center of Pressure Report* (p. 1515) for information about matching additional characters using * and ?.

Save Output Parameter...
opens the *Save Output Parameter Dialog Box* (p. 2044).

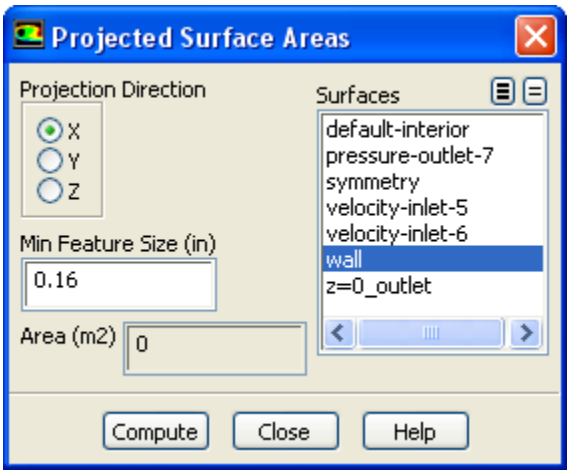
Print
displays (in the console) the pressure, viscous (if appropriate), and total forces or moments, and the pressure, viscous, and total force coefficients along the specified force vector or moment center for the selected wall zones. The center of pressure coordinates will print to the console when the **Center of Pressure** option is activated.

Write...
opens *The **Select File** Dialog Box* (p. 33), which you can use to save the reported values to a file.

35.21.3. Projected Surface Areas Dialog Box

The **Projected Surface Areas** dialog box allows you to compute an estimated area of the projection of selected surfaces along the x, y, or z axis. See *Projected Surface Area Calculations* (p. 1518) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Projection Direction
indicates the direction along which to project the surface.

Min Feature Size
specifies the length of the smallest feature in the geometry that you want to resolve in the area calculation.

Area
displays the computed projected area.

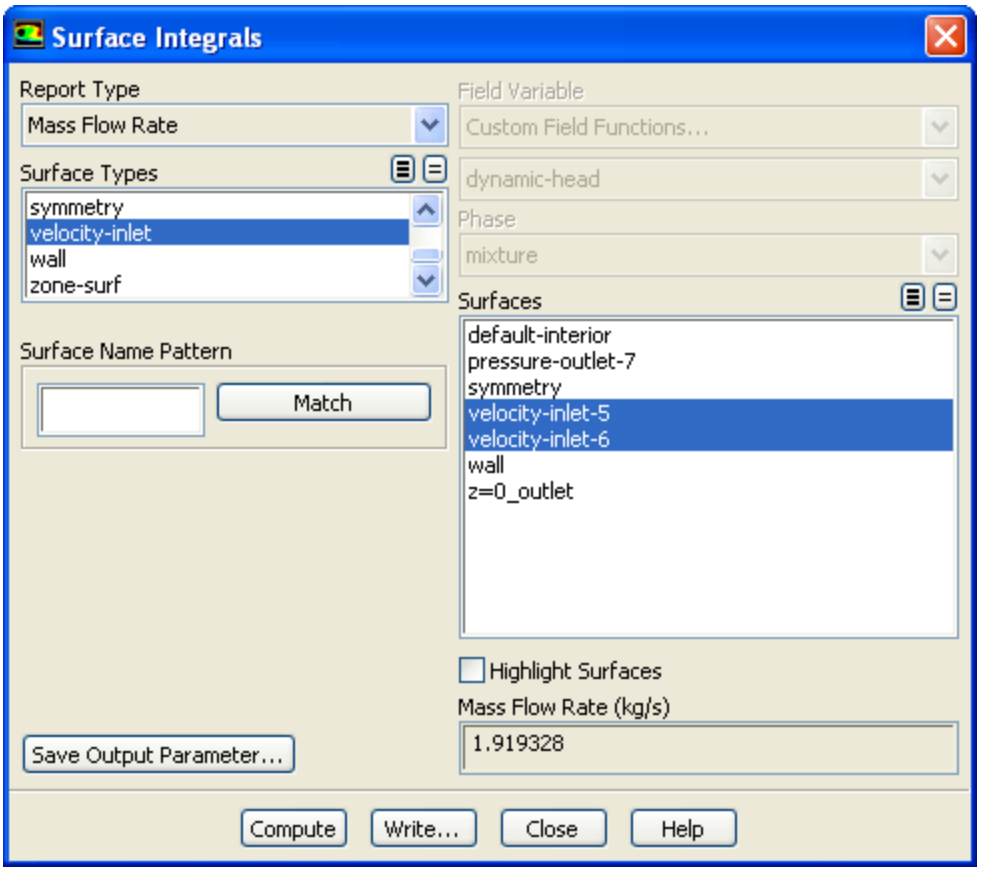
Surfaces
contains a list of existing surfaces. You can select the surface(s) for which the projected area is to be calculated in this list.

Compute
computes the area of the selected surfaces projected along the selected direction. The area will be printed in the **Area** box and in the console window.

35.21.4. Surface Integrals Dialog Box

The **Surface Integrals** dialog box allows you to compute the area, mass and volume flow rate, standard deviation, integral, flow rate, area-weighted average, mass-weighted average, sum, facet average, facet minimum/maximum, vertex average, or vertex minimum/maximum quantity of a specified field variable on a selected list of surfaces. See *Surface Integration* (p. 1519) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Type

contains drop-down list that has options that control the method of surface integration.

Area

turns on the computation of the surface area.

Area-Weighted Average

turns on the computation of the area-weighted average on the surface(s).

Facet Average

turns on the computation of the facet-averaged quantity on the surface(s).

Facet Minimum

turns on the computation of the facet minimum of a quantity on the surface(s).

Facet Maximum

turns on the computation of the facet maximum of a quantity on the surface(s).

Flow Rate

turns on the computation of the flow rate through the surface(s).

Integral

turns on the computation of the integral on the surface(s).

Mass Flow Rate

turns on the computation of the mass flow rate through the surface(s).

Mass-Weighted Average

turns on the computation of the mass-averaged quantity on the surface(s).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard Deviation
turns on the computation of the standard deviation of a specified field variable on a surface.

Sum
turns on the computation of the summed quantity on the surface(s).

Vertex Average
turns on the computation of the vertex-averaged quantity on the surface(s).

Vertex Minimum
turns on the computation of the vertex minimum of a quantity on the surface(s).

Vertex Maximum
turns on the computation of the vertex maximum of a quantity on the surface(s).

Volume Flow Rate
turns on the computation of the volume flow rate through the surface(s).

Surface Types
contains a selectable list of types of surfaces. If you select (or deselect) an item in this list, all surfaces of that type will be selected (or deselected) automatically in the **Surfaces** list.

Surface Name Pattern
specifies the pattern to look for in the names of surfaces. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Surfaces** list with names that match the specified pattern. See *Generating a Surface Integral Report* (p. 1519) for information about matching additional characters using * and ?.

Field Variable
contains a list of the field variables that can be used in the surface integrations. This option is not active if the **Area**, **Mass Flow Rate**, or **Volume Flow Rate** option is active.

Surfaces
is a selectable list of surfaces.

Area
displays the result of the area summation over all the selected surfaces. This label is visible when **Area** is active.

Integral
displays the result of the integral computation over all the selected surfaces. This label is visible when **Integral** is active.

Standard Deviation
displays the result of the standard deviation computation over all the selected surfaces. This label is visible when **Standard Deviation** is active.

Flow Rate
displays the result of the flow rate computation over all the selected surfaces. This label is visible when **Flow Rate** is active.

Mass Flow Rate
displays the result of the mass flow rate computation over all the selected surfaces. This label is visible when **Mass Flow Rate** is active.

Volumetric Flow Rate
displays the result of the volumetric flow rate computation over all the selected surfaces. This label is visible when **Volume Flow Rate** is active.

Area-Weighted Average
displays the result of the area-weighted average computation over all the selected surfaces. This label is visible when **Area-Weighted Average** is active.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass-Weighted Average
displays the result of the mass-averaged computation over all the selected surfaces. This label is visible when **Mass-Weighted Average** is active.

Sum of Facet Values
displays the result of the summation over all the selected surfaces. This label is visible when **Sum** is active.

Average of Facet Values
displays the result of the facet-averaged computation over all the selected surfaces. This label is visible when **Facet Average** is active.

Minimum of Facet Values
displays the minimum facet value on all the selected surfaces. This label is visible when **Facet Minimum** is active.

Maximum of Facet Values
displays the maximum facet value on all the selected surfaces. This label is visible when **Facet Maximum** is active.

Average of Surface Vertex Values
displays the result of the vertex-averaged computation over all the selected surfaces. This label is visible when **Vertex Average** is active.

Minimum of Vertex Values
displays the minimum facet value on all the selected surfaces. This label is visible when **Vertex Minimum** is active.

Maximum of Vertex Values
displays the maximum facet value on all the selected surfaces. This label is visible when **Vertex Maximum** is active.

Highlight Surfaces
when enabled highlights the surfaces (selected in the **Surface Integrals** dialog box) in the graphics window.

Save Output Parameter...
opens the *Save Output Parameter Dialog Box* (p. 2044).

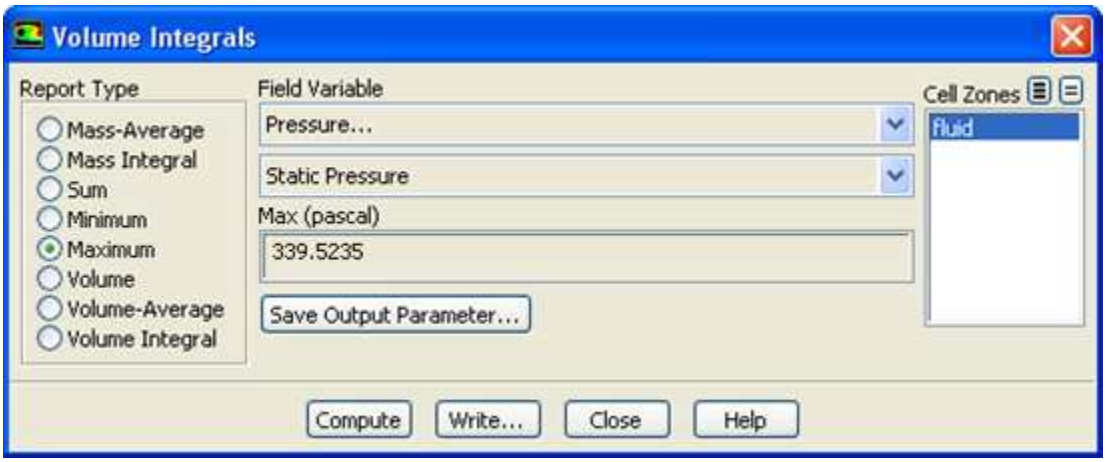
Compute
computes the specified integration on the selected surfaces.

Write...
opens *The **Select File** Dialog Box* (p. 33), which you can use to save the reported values to a file.

35.21.5. Volume Integrals Dialog Box

The **Volume Integrals** dialog box allows you to compute the volume, sum, maximum, minimum, volume integral, volume-averaged quantity, mass integral, or mass-averaged quantity of a specified field variable on a selected list of cell zones. See *Volume Integration* (p. 1521) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Type

contains radio buttons that control the method of volume integration.

Mass-Average

turns on the computation of the mass-averaged quantity on the cell zone.

Mass Integral

turns on the computation of the mass integral on the cell zone.

Sum

turns on the computation of the summation over all cells in the selected zone.

Minimum

computes the minimum value of the selected variable at each cell in the selected zone. selected zone.

Maximum

computes the maximum value of the selected variable at each cell in the selected zone. selected zone.

Volume

turns on the computation of the cell zone volume.

Volume Integral

turns on the computation of the volume integral on the cell zone.

Volume-Average

turns on the computation of the volume-weighted average on the cell zone.

Field Variable

contains a list of the field variables that can be used in the sum, volume integral, and average computations. This option is not active if the **Volume** option is active.

Cell Zones

is a selectable list of cell zones.

Total Volume

displays the result of the volume computation over all the selected zones. This label is visible when **Volume** is active.

Sum

displays the result of the summation over all the selected zones. This label is visible when **Sum** is active.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max
displays the result of the maximum value of the selected zone(s). This label is visible when **Maximum** is active.

Min
displays the result of the minimum value of the selected zone(s). This label is visible when **Minimum** is active.

Total Volume Integral
displays the result of the volume-integral computation over all the selected zones. This label is visible when **Volume Integral** is active.

Volume-Weighted Average
displays the result of the volume-averaged computation over all the selected zones. This label is visible when **Volume-Average** is active.

Total Mass-Weighted Integral
displays the result of the mass-integral computation over all the selected zones. This label is visible when **Mass Integral** is active.

Mass-Weighted Average
displays the result of the mass-averaged computation over all the selected zones. This label is visible when **Mass-Average** is active.

Save Output Parameter...
opens the *Save Output Parameter Dialog Box* (p. 2044).

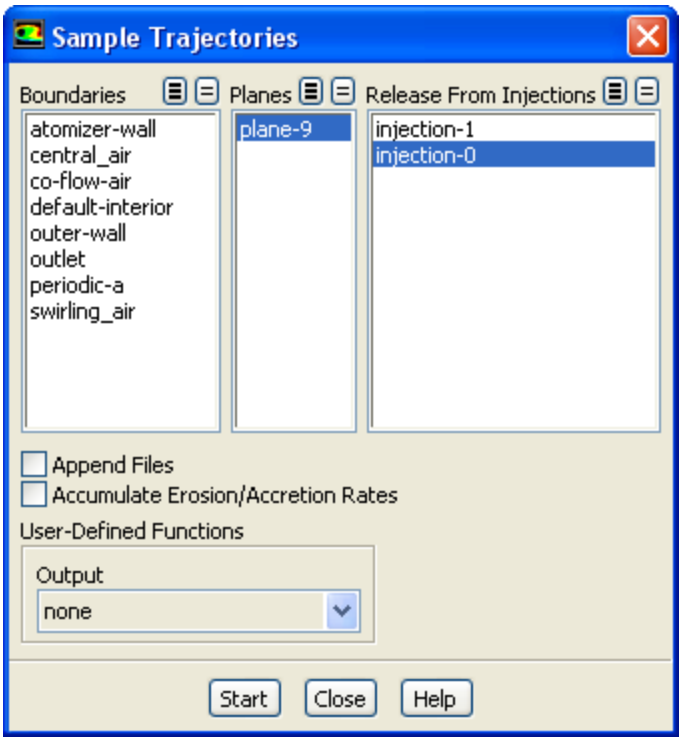
Compute
computes the specified integration on the selected zones.

Write...
opens *The **Select File** Dialog Box* (p. 33), which you can use to save the reported values to a file.

35.21.6. Sample Trajectories Dialog Box

The **Sample Trajectories** dialog box allows the writing of particle states (position, velocity, diameter, temperature, and mass flow rate) at various boundaries and planes (lines in 2D). See *Sampling of Trajectories* (p. 1082) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Boundaries
lists boundaries that can be chosen as the surfaces at which samples will be written.

Lines
(in 2D) lists lines that can be chosen as the surfaces at which samples will be written.

Planes
(in 3D) lists planes that can be chosen as the surfaces at which samples will be written.

Release From Injections
lists injections from which the injection to be tracked is chosen.

Append Files
causes the results of multiple calculations to be appended to a single file.

Accumulate Erosion/Accretion Rates
causes erosion and accretion rates to be accumulated for repeated trajectory calculations.

User-Defined Functions
allow control of the format and the information written for the sample output.

Output
contains a drop-down list of available user-defined functions.

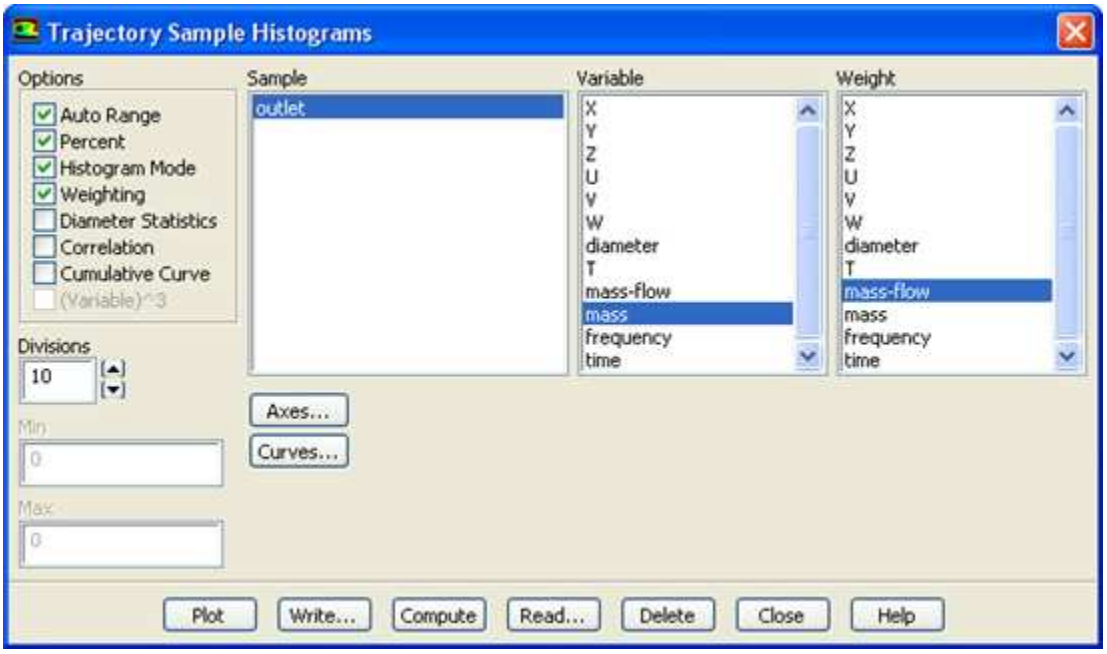
Compute
causes the particles to be tracked and their status to be written to files when they encounter selected surfaces. This button will not appear for unsteady particle tracking.

Start
initiates sampling for unsteady particle tracking. This button will replace the **Compute** button if you are performing unsteady particle tracking. After you click it, it will change to the **Stop** button. Click **Stop** to stop the sampling. (The label will change back to **Start**.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.21.7. Trajectory Sample Histograms Dialog Box

The **Trajectory Sample Histograms** dialog box allows the plotting of histograms from sample files created in the *Sample Trajectories Dialog Box* (p. 2036). See *Histogram Reporting of Samples* (p. 1084) for details about the items below.



Controls

Options

contains check buttons for histogram options.

Auto Range

toggles between automatic and manual settings of the histogram range.

Percent

causes the plot to indicate the percent of particles. Deselecting this will result in the actual number of particles being plotted.

Histogram Mode

allows you to display the histogram with or without bars.

Weighting

allows you to apply a weight to the data sample.

Diameter Statistics

allows you to display a summary of diameter statistics for a selected variable in the console window.

Correlation

allows you to choose correlations between how one particle variable depends on another particle variable.

Cumulative Curve

allows you to compute a cumulative distribution curve for a selected variable.

(Variable^3)

allows you to plot the cumulative mass distribution for a constant particle density using the particle diameter.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Divisions
sets the number of "bins" in the histogram.

Sample
lists the data samples that have been read in.

Variable
lists the fields variables available in the selected sample.

Weight
lists the weighted fields variables available in the selected sample.

Correlation
lists the sampled variables, allowing you to choose the correlation variable.

Min
displays the minimum value of the variable selected in the **Variable** list. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
displays the maximum value of the variable selected in the **Fields** list. If **Auto Range** is off, you can set the maximum by typing a new value.

Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the histogram axes.

Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the histogram.

Plot
displays the histogram in the active graphics window.

Write...
allows you store the histograms in an XY-plot file format

Compute
updates the **Min** and **Max** values.

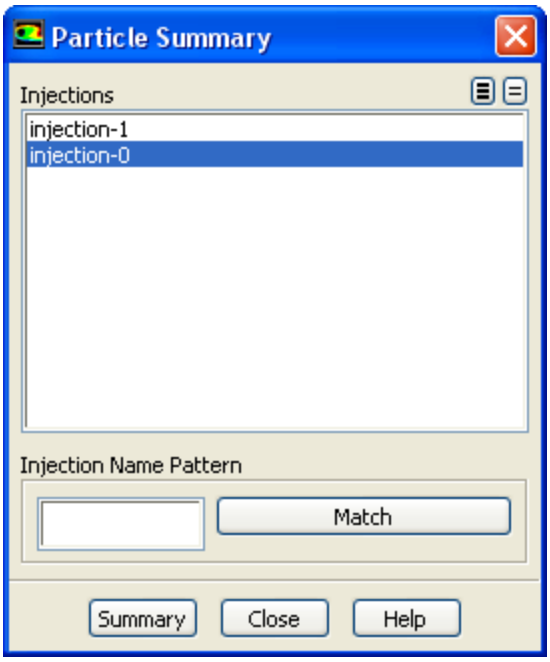
Read...
opens *The **Select File** Dialog Box* (p. 33), where you can select a sample file to be read.

Delete
removes the sample selected in the **Sample** list.

35.21.8. Particle Summary Dialog Box

The **Particle Summary** dialog box allows you to report a summary for particle injections. See *Summary Reporting of Current Particles* (p. 1085) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

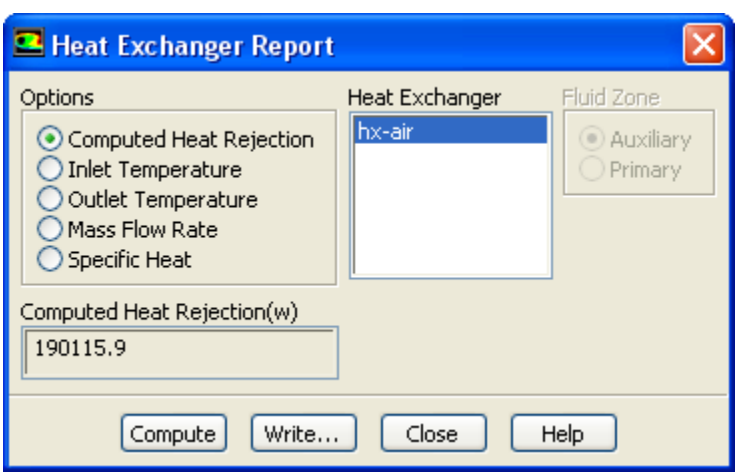
Injections
lists the particle injection(s) for which you can generate a summary.

Injection Name Pattern
specifies the pattern to look for in the names of injections. Type the pattern in the text field and click **Match** to select (or deselect) the injection in the **Injections** list with names that match the specified pattern. See *Summary Reporting of Current Particles* (p. 1085) for information about matching additional characters using * and ?.

Summary
prints the injection summary in the console window.

35.21.9. Heat Exchanger Report Dialog Box

The **Heat Exchanger Report** dialog box allows you to report a summary for heat exchangers. See *Postprocessing for the Heat Exchanger Model* (p. 794) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

lists the available reporting options for the heat exchanger.

Computed Heat Rejection

allows you to report the heat rejection calculated from the heat exchanger.

Inlet Temperature

allows you to report the inlet temperature for both the primary and the auxiliary heat exchanger fluid.

Outlet Temperature

allows you to report the outlet temperature for both the primary and the auxiliary heat exchanger fluid.

Mass Flow Rate

allows you to report the mass flow rate for both the primary and the auxiliary heat exchanger fluid.

Specific Heat

allows you to report the specific heat for both the primary and the auxiliary heat exchanger fluid.

Result

displays the results of the calculations once the **Compute** button is selected.

Heat Exchanger

displays a list of heat exchanger fluid zones

Fluid Zone

(not available when **Computed Heat Rejection** option is selected.) allows you to report quantities for either the primary or auxiliary fluid zones.

Compute

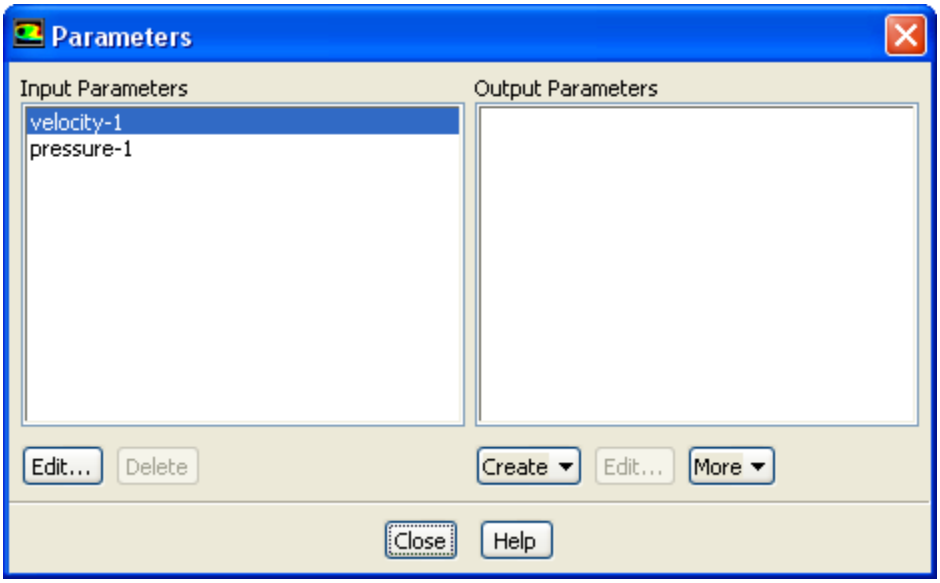
computes the specified quantity on the selected zones.

Write...

opens *The **Select File** Dialog Box* (p. 33), which you can use to save the reported values to a file.

35.21.10. Parameters Dialog Box

The **Parameters** dialog box allows you to create output parameters, which allow you to compare reporting values for different cases. See *Creating Output Parameters* (p. 1509) for details about the items below.



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Input Parameters
contains a list of existing input parameters. See *Defining and Viewing Parameters* (p. 210) for details about creating input parameters.

Edit...
opens the *Input Parameter Properties Dialog Box* (p. 2043).

Important

When using ANSYS FLUENT in ANSYS Workbench, parameters are not editable, so the **Edit...** button becomes the **View...** button. This opens the *Input Parameter Properties Dialog Box* (p. 2043) where the parameter properties can only be viewed. For more information, see the separate ANSYS FLUENT in **Workbench** User's Guide.

Delete
removes the selected input parameter from the list of **Input Parameters**.

Output Parameters
contains a list of existing output parameters.

Create
contains a drop-down list that allows you to create additional output parameters. Available options include:

Fluxes
opens the *Flux Reports Dialog Box* (p. 2027).

Forces
opens the *Force Reports Dialog Box* (p. 2029).

Surface Integrals
opens the *Surface Integrals Dialog Box* (p. 2031).

Volume Integrals
opens the *Volume Integrals Dialog Box* (p. 2034).

Edit...
opens the dialog box corresponding to the selected item in the **Output Parameters** list.

More
contains a drop-down list containing additional tasks that you can perform, including:

Delete
displays a message in a dialog box, prompting you for a response to confirm the deletion of the output parameter.

Rename
allows you to edit the name of the output parameter through the *Rename Dialog Box* (p. 2043).

Print to Console
reports values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

Print All to Console
outputs the values from all output parameters to the console window.

Write...
allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write All...
prompts you for a file name and then writes the values for all of the output parameters to a file.

35.21.11. Rename Dialog Box

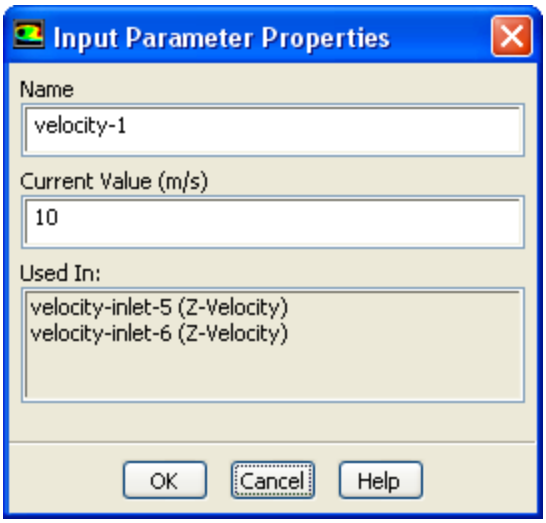
The **Rename** dialog box allows you to change the name of the output parameter that you created.

Controls

New Name
contains the new name of the output parameter.

35.21.12. Input Parameter Properties Dialog Box

The **Input Parameter Properties** dialog box allows you to create input parameters, which allow you to compare reporting values for different cases. See *Creating Output Parameters* (p. 1509) for details about the items below.



Controls

Name
contains the name of the input parameter.

Current Value
contains the current value of the input parameter.

Used In
indicates the cell zone or boundary condition where the input parameter is currently used.

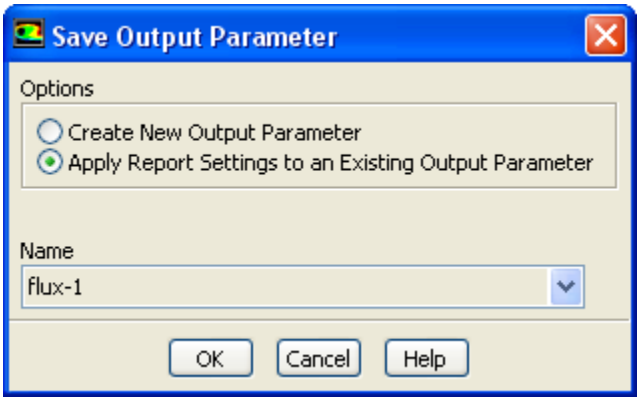
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

Source term parameters can only be specified using SI units. The default parameter name will not include the unit quantity name. Even if you have selected units other than SI for the source term parameters, you must still provide values assuming SI units for the (source-term) parameters in *both* FLUENT and Workbench.

35.21.13. Save Output Parameter Dialog Box

The **Save Output Parameter** dialog box allows you to save specific output parameters which allow you to compare reporting values for different cases. See *Creating Output Parameters* (p. 1509) for details about the items below.



Controls

Options

contains options for saving your output parameters.

Create New Output Parameter

allows you to create a new output parameter.

Apply Report Settings to an Existing Output Parameter

allows you to overwrite an existing output parameter of the same type.

Name

contains the name for the current output parameter.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 36: Menu Reference Guide

This reference guide provides information about the menus in FLUENT

36.1. File Menu
36.2. Mesh Menu
36.3. Define Menu
36.4. Solve Menu
36.5. Adapt Menu
36.6. Surface Menu
36.7. Display Menu
36.8. Report Menu
36.9. Parallel Menu
36.10. View Menu
36.11. Turbo Menu
36.12. Help Menu

36.1. File Menu

For additional information, please see the following sections:

36.1.1. File/Read/Mesh...
36.1.2. File/Read/Case...
36.1.3. File/Read/Data...
36.1.4. File/Read/Case & Data...
36.1.5. File/Read/PDF...
36.1.6. File/Read/ISAT Table...
36.1.7. File/Read/DTRM Rays...
36.1.8. File/Read/View Factors...
36.1.9. File/Read/Profile...
36.1.10. File/Read/Scheme...
36.1.11. File/Read/Journal...
36.1.12. File/Write/Case...
36.1.13. File/Write/Data...
36.1.14. File/Write/Case & Data...
36.1.15. File/Write/PDF...
36.1.16. File/Write/ISAT Table...
36.1.17. File/Write/Flamelet...
36.1.18. File/Write/Surface Clusters...
36.1.19. File/Write/Profile...
36.1.20. File/Write/Autosave...
36.1.21. File/Write/Boundary Mesh...
36.1.22. File/Write/Start Journal...
36.1.23. File/Write/Stop Journal
36.1.24. File/Write/Start Transcript...
36.1.25. File/Write/Stop Transcript
36.1.26. File/Import/ABAQUS/Input File...
36.1.27. File/Import/ABAQUS/Filbin File...
36.1.28. File/Import/ABAQUS/ODB File...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.29. File/Import/CFX/Definition File...
36.1.30. File/Import/CFX/Result File...
36.1.31. File/Import/CGNS/Mesh...
36.1.32. File/Import/CGNS/Data...
36.1.33. File/Import/CGNS/Mesh & Data...
36.1.34. File/Import/EnSight...
36.1.35. File/Import/FIDAP...
36.1.36. File/Import/GAMBIT...
36.1.37. File/Import/HYPERMESH ASCII...
36.1.38. File/Import/IC3M...
36.1.39. File/Import/I-deas Universal...
36.1.40. File/Import/LSTC/Input File...
36.1.41. File/Import/LSTC/State File...
36.1.42. File/Import/Marc POST...
36.1.43. File/Import/Mechanical APDL/Input File...
36.1.44. File/Import/Mechanical APDL/Result File...
36.1.45. File/Import/NASTRAN/Bulkdata File...
36.1.46. File/Import/NASTRAN/Op2 File...
36.1.47. File/Import/PATRAN/Neutral File...
36.1.48. File/Import/PATRAN/Result File...
36.1.49. File/Import/PLOT3D/Grid File...
36.1.50. File/Import/PLOT3D/Result File...
36.1.51. File/Import/PTC Mechanica Design...
36.1.52. File/Import/Tecplot...
36.1.53. File/Import/FLUENT 4 Case File...
36.1.54. File/Import/PreBFC File...
36.1.55. File/Import/Partition/Metis...
36.1.56. File/Import/Partition/Metis Zone...
36.1.57. File/Import/CHEMKIN Mechanism...
36.1.58. File/Export/Solution Data..
36.1.59. File/Export/Particle History Data..
36.1.60. File/Export/During Calculation/Solution Data..
36.1.61. File/Export/During Calculation/Particle History Data..
36.1.62. File/Export to CFD-Post...
36.1.63. File/Solution Files...
36.1.64. File/Interpolate...
36.1.65. File/EM Mapping/HFSS...
36.1.66. File/EM Mapping/Maxwell...
36.1.67. File/EM Mapping/Q3D...
36.1.68. File/FSI Mapping/Volume...
36.1.69. File/FSI Mapping/Surface...
36.1.70. File/Save Picture...
36.1.71. File/Data File Quantities...
36.1.72. File/Batch Options...
36.1.73. File/RSF...
36.1.74. File/Exit

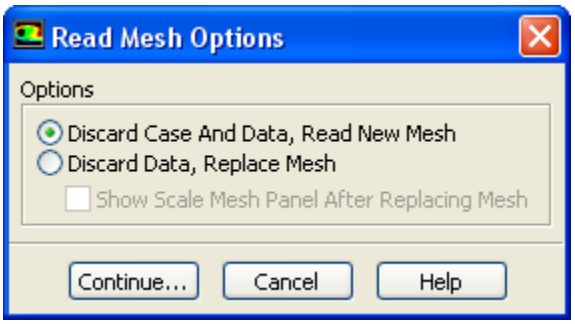
36.1.1. File/Read/Mesh...

The **File/Read/Mesh...** menu item opens the *Read Mesh Options Dialog Box* (p. 2047) which allows you to read or replace a mesh.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.1.1. Read Mesh Options Dialog Box

The **Read Mesh Options** dialog box is used to read or replace a mesh. See *Reading Mesh Files* (p. 62) for more information.



Controls

Options

allows you to read in a new mesh or replace the existing mesh.

Discard Case And Data, Read New Mesh

results in both the case and data files being discarded when reading in a new mesh.

Discard Data, Replace Mesh

results in the data file being discarded when replacing an existing mesh.

Show Scale Mesh Panel After Replacing Mesh

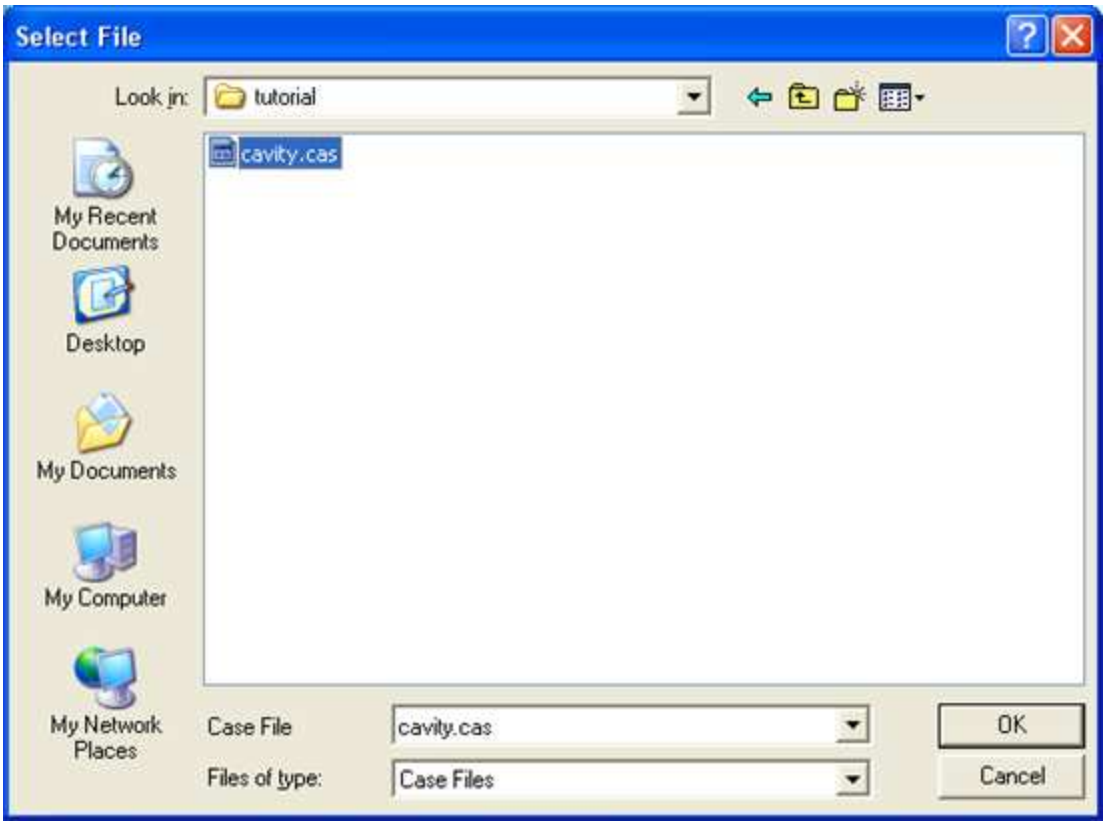
gives you the option to have the *Scale Mesh Dialog Box* (p. 1634) appear automatically for you to check or scale your mesh.

36.1.2. File/Read/Case...

The **File/Read/Case...** menu item is used to read in an ANSYS FLUENT case file (extension `.cas`), or a mesh file (extension `.msh`, `.grd`, `.MSH`, or `.GRD`) that has been saved in the native format for ANSYS FLUENT. See *Reading Mesh Files* (p. 62) and *Reading and Writing Case and Data Files* (p. 63) for details.

The **File/Read/Case...** menu item opens *The **Select File** Dialog Box* (p. 33) which allows you to select the appropriate file to be read.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



36.1.3. File/Read/Data...

The **File/Read/Data...** menu item is used to read in an ANSYS FLUENT data file (which has a `.dat` extension) or parallel data file (which has a `.pdat` extension). This menu item will not be available until you read in a case or mesh file. See *Reading and Writing Data Files* (p. 64) and *Reading and Writing Parallel Data Files* (p. 68) for details.

The **File/Read/Data...** menu item opens *The **Select File** Dialog Box* (p. 33) which allows you to select the appropriate file to be read.

36.1.4. File/Read/Case & Data...

The **File/Read/Case & Data...** menu item is used to read in an ANSYS FLUENT case file and the corresponding data file (e.g., `myfile.cas` and `myfile.dat`) together. See *Reading and Writing Case and Data Files Together* (p. 65) for details.

The **File/Read/Case & Data...** menu item opens *The **Select File** Dialog Box* (p. 33) which allows you to select the appropriate files to be read. Select the appropriate case file, and the corresponding data file (i.e., the file having the same name with a `.dat` extension) will also be read in.

36.1.5. File/Read/PDF...

The **File/Read/PDF...** menu item is used to read a PDF file (extension `.pdf`) created by ANSYS FLUENT for use with the non-premixed or partially premixed combustion model. This menu item is available only when the non-premixed or partially premixed combustion model has been enabled. See *Saving the Look-Up Tables* (p. 874) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.6. File/Read/ISAT Table...

The **File/Read/ISAT Table...** menu item is used to read an ISAT table (extension `.isat` for use with the Composition PDF Transport model. This menu item is available only when the Composition PDF Transport model has been enabled. See *Using ISAT Efficiently* (p. 925) for details.

36.1.7. File/Read/DTRM Rays...

The **File/Read/DTRM Rays...** menu item is used to read a ray file (extension `.ray`) created by ANSYS FLUENT for use with the DTRM (radiation model). This menu item is available only when the DTRM has been enabled. See *Writing and Reading the DTRM Ray File* (p. 709) for details.

36.1.8. File/Read/View Factors...

The **File/Read/View Factors...** menu item is used to read in a view factor file for use with the surface-to-surface (S2S) radiation model. This menu item is available only when the S2S model has been enabled. See *Reading View Factors into ANSYS FLUENT* (p. 720) for details.

36.1.9. File/Read/Profile...

The **File/Read/Profile...** menu item opens the **Select File** dialog for reading profiles (*Reading Profile Files* (p. 70)), It is used to read a cell zone or boundary condition profile file (extension `.prof`). A profile file defines profiles that can be used to specify flow conditions for a cell zone or a boundary. See *Profiles* (p. 366) for details.

36.1.10. File/Read/Scheme...

The **File/Read/Scheme...** menu item is used to read in a Scheme source file (extension `.scm`). See *Reading Scheme Source Files* (p. 73) for details.

36.1.11. File/Read/Journal...

The **File/Read/Journal...** menu item is used to read in a journal file (extension `.jou`) containing a sequence of ANSYS FLUENT commands. You can create a journal file using the **File/Write/Journal...** menu item. See *Creating and Reading Journal Files* (p. 73) for details.

36.1.12. File/Write/Case...

The **File/Write/Case...** menu item is used to save an ANSYS FLUENT case file. See *Reading and Writing Case Files* (p. 64) for details.

The **File/Write/Case...** menu item opens *The **Select File** Dialog Box* (p. 33) which allows you to save the file with a name of choice. The dialog box is similar to the **Select File** dialog for reading files, except that it has an additional option for writing binary files.

Important

- When ANSYS FLUENT writes a case file, the `.cas` extension is added to the file name specified, unless the name already ends with `.cas`.
- You can also write a compressed file by appending `.gz` or `.Z` to the file name. (See *Reading and Writing Compressed Files* (p. 59) for details about file compression.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.13. File/Write/Data...

The **File/Write/Data...** menu item is used to save an ANSYS FLUENT data file (which has a `.dat` extension) or parallel data file (which has a `.pdat` extension). See *Reading and Writing Data Files* (p. 64) and *Reading and Writing Parallel Data Files* (p. 68) for details.

Important

- When ANSYS FLUENT writes a data file, the `.dat` extension is added to the file name specified, unless the name already ends with `.dat` or `.pdat`.
- You can compress data files by appending `.gz` or `.Z` to the file name (see *Reading and Writing Compressed Files* (p. 59) for details about file compression). To compress parallel data files (i.e., files saved with a `.pdat` extension), you must use asynchronous file compression (see *Reading and Writing Case and Data Files* (p. 63)).

36.1.14. File/Write/Case & Data...

The **File/Write/Case & Data...** menu item is used to save an ANSYS FLUENT case file and data file at the same time (e.g., `myfile.cas` and `myfile.dat`). See *Reading and Writing Case and Data Files Together* (p. 65) for details.

Enter the name of the case file in the text entry box in the **Select File** dialog box and the corresponding data file (same file name, but with a `.dat` extension) will also be written.

36.1.15. File/Write/PDF...

The **File/Write/PDF...** menu item is used to write a PDF file after computing the look-up tables using the non-premixed or partially premixed combustion model in ANSYS FLUENT. This menu item is available only when the non-premixed or partially premixed combustion model has been enabled. See *Saving the Look-Up Tables* (p. 874) for details.

36.1.16. File/Write/ISAT Table...

The **File/Write/ISAT Table...** menu item is used to write an ISAT table. See *Using ISAT Efficiently* (p. 925) for details.

36.1.17. File/Write/Flamelet...

The **File/Write/Flamelet...** menu item is used to write a flamelet file generated using the non-premixed or partially premixed combustion model in ANSYS FLUENT. This menu item is available only when the non-premixed or partially premixed combustion model has been enabled. See The Laminar Flamelet Models Theory in the Theory Guide for details.

36.1.18. File/Write/Surface Clusters...

The **File/Write/Surface Clusters...** menu item is used to set parameters related to surface clusters and view factors for the surface-to-surface radiation model. It opens the *View Factors and Clustering Dialog Box* (p. 1660). The dialog box that you open using the **File/Write/Surface Clusters...** menu item is different than the one opened via the **Settings...** button in the **Radiation Model** dialog box, in that it saves the settings you specify to a file, which can be used to calculate the view factors outside of ANSYS FLUENT (see *Computing View*

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Factors Outside ANSYS FLUENT (p. 718)). When you click **OK**, the *The **Select File** Dialog Box* (p. 33) will open so that you can specify a name for the file where ANSYS FLUENT should save the settings.

36.1.19. File/Write/Profile...

The **File/Write/Profile...** menu item opens the *Write Profile Dialog Box* (p. 1811).

36.1.20. File/Write/Autosave...

The **File/Write/Autosave...** menu item opens the *Autosave Dialog Box* (p. 1943).

36.1.21. File/Write/Boundary Mesh...

The **File/Write/Boundary Mesh...** menu item is used to write the boundary zones (surface mesh) of the domain to a file. You can then read this file into GAMBIT or TGrid to produce an improved volume mesh. See *Writing a Boundary Mesh* (p. 72) for details.

36.1.22. File/Write/Start Journal...

The **File/Write/Start Journal...** menu item is used to start the recording of subsequent ANSYS FLUENT commands to a journal file. You can read this journal file back into the solver later (using the **File/Read/Journal...** menu item) to automate the execution of the recorded commands. See *Creating and Reading Journal Files* (p. 73) for details.

36.1.23. File/Write/Stop Journal

The **File/Write/Stop Journal** menu item replaces the **File/Write/Start Journal...** menu item after the recording of a journal file has begun. The **File/Write/Stop Journal** menu item is used to end the journal recording. See *Creating and Reading Journal Files* (p. 73) for details.

36.1.24. File/Write/Start Transcript...

The **File/Write/Start Transcript...** menu item is used to start the recording of a transcript file containing all input to and output from the solver. (You cannot read a transcript file back into the solver.) See *Creating Transcript Files* (p. 75) for details.

36.1.25. File/Write/Stop Transcript

The **File/Write/Stop Transcript** menu item replaces the **File/Write/Start Transcript...** menu item after the recording of a transcript file has begun. The **File/Write/Stop Transcript** menu item is used to end the transcript recording. See *Creating Transcript Files* (p. 75) for details.

36.1.26. File/Import/ABAQUS/Input File...

The **File/Import/ABAQUS/Input File...** menu item is used to import an ABAQUS input file (extension `.inp`) which contains the input description of a finite element model for the ABAQUS finite element program. See *ABAQUS Files* (p. 77) for details.

36.1.27. File/Import/ABAQUS/Filbin File...

The **File/Import/ABAQUS/Filbin File...** menu item is used to import an ABAQUS filbin file (extension `.fil`) which contains the finite element model and results data. See *ABAQUS Files* (p. 77) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.28. File/Import/ABAQUS/ODB File...

The **File/Import/ABAQUS/ODB File...** menu item is used to import an ABAQUS ODB file with a `.odb` extension. See *ABAQUS Files* (p. 77) for details.

36.1.29. File/Import/CFX/Definition File...

The **File/Import/CFX/Definition File...** menu item is used to import a CFX definition file (extension `.def`) which contains mesh information to be read into the solver. See *ANSYS CFX Files* (p. 77) for details.

36.1.30. File/Import/CFX/Result File...

The **File/Import/CFX/Result File...** menu item is used to import a CFX result file (extension `.res`). See *ANSYS CFX Files* (p. 77) for details.

36.1.31. File/Import/CGNS/Mesh...

The **File/Import/CGNS/Mesh...** menu item is used to read in a CGNS-format mesh file (extension `.cgns`). See *Meshes and Data in CGNS Format* (p. 78) for details.

36.1.32. File/Import/CGNS/Data...

The **File/Import/CGNS/Data...** menu item is used to read in a CGNS-format data file. See *Meshes and Data in CGNS Format* (p. 78) for details.

36.1.33. File/Import/CGNS/Mesh & Data...

The **File/Import/CGNS/Mesh & Data...** menu item is used to read in a set of CGNS-format mesh and data files. See *Meshes and Data in CGNS Format* (p. 78) for details.

36.1.34. File/Import/EnSight...

The **File/Import/EnSight...** menu item is used to import EnSight files (extension `.encas` or `.case`). See *EnSight Files* (p. 78) for details.

36.1.35. File/Import/FIDAP...

The **File/Import/FIDAP...** menu item is used to import an ANSYS FIDAP neutral file (extension `.FDNEUT` or `.unv`). See *ANSYS FIDAP Neutral Files* (p. 79) for details.

36.1.36. File/Import/GAMBIT...

The **File/Import/GAMBIT...** menu item is used to read in a neutral file from GAMBIT. See *GAMBIT and GeoMesh Mesh Files* (p. 79) for details.

36.1.37. File/Import/HYPERMESH ASCII...

The **File/Import/HYPERMESH ASCII...** menu item is used to import a **HYPERMESH ASCII** file (extension `.hm`, `.hma`, or `.hmascii`). See *HYPERMESH ASCII Files* (p. 79) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.38. File/Import/IC3M...

The **File/Import/IC3M...** menu item is used to import files created using ICEM CFD IC3M, in order to automatically set up a case file for a dynamic mesh simulation of an in-cylinder problem. This menu item is only available when running the 3D version of ANSYS FLUENT. See *IC3M Files* (p. 79) for details.

Important

IC3M files should not be used with ANSYS FLUENT in Workbench.

36.1.39. File/Import/I-deas Universal...

The **File/Import/I-deas Universal...** menu item is used to import an I-deas Universal file which contains mesh information and zone types to be read into the solver. See *I-deas Universal Files* (p. 80) for details.

36.1.40. File/Import/LSTC/Input File...

The **File/Import/LSTC/Input File...** menu item is used to import an LSTC input file (extension `.k`, `.key`, or `.dyn`) which contains the input description of a finite element model for the LS-DYNA finite element program. See *LSTC Files* (p. 80) for details.

36.1.41. File/Import/LSTC/State File...

The **File/Import/LSTC/State File...** menu item is used to import an LSTC state file (extension `.d3plot`) which contains control data, geometry data, and state data. See *LSTC Files* (p. 80) for details.

36.1.42. File/Import/Marc POST...

The **File/Import/Marc POST...** menu item is used to import a Marc POST file generated using the MSC Marc finite element program. See *Marc POST Files* (p. 81) for details

36.1.43. File/Import/Mechanical APDL/Input File...

The **File/Import/Mechanical APDL/Input File...** menu item is used to import a Mechanical APDL input file (extensions `.ans`, `neu`, `.cdb`, or `.prep7`) which contains mesh information to be read into the solver. See *Mechanical APDL Files* (p. 81) for details.

36.1.44. File/Import/Mechanical APDL/Result File...

The **File/Import/Mechanical APDL/Result File...** menu item is used to import a Mechanical APDL result file (extension `.rfl`, `.rst`, `.rth`, or `.rmg`). See *Mechanical APDL Files* (p. 81) for details.

36.1.45. File/Import/NASTRAN/Bulkdata File...

The **File/Import/NASTRAN/Bulkdata File...** menu item is used to import a NASTRAN Bullkdata file (extension `.nas`, `.dat`, or `.bdf`) which contains mesh information to be read into the solver. See *NASTRAN Files* (p. 81) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.46. File/Import/NASTRAN/Op2 File...

The **File/Import/NASTRAN/Op2 File...** menu item is used to import a NASTRAN Op2 file (extension `.op2`) which is an output binary data file containing data used in the NASTRAN finite element program. See *NASTRAN Files* (p. 81) for details.

36.1.47. File/Import/PATRAN/Neutral File...

The **File/Import/PATRAN/Neutral File...** menu item is used to read in a PATRAN Neutral file (extension `.neu`, `.out`, or `.pat`) zoned by named components (i.e., with the nodes placed into zones based on group name). The PATRAN neutral file contains mesh information to be read into the solver. See *PATRAN Neutral Files* (p. 82) for details.

36.1.48. File/Import/PATRAN/Result File...

The **File/Import/PATRAN/Result File...** menu item is used to import a PATRAN result file. See *PATRAN Neutral Files* (p. 82) for details.

36.1.49. File/Import/PLOT3D/Grid File...

The **File/Import/PLOT3D/Grid File...** menu item is used to import a PLOT3D grid file (extension `.g`, `.x`, `.xyz`, or `.grd`). See *PLOT3D Files* (p. 82) for details.

36.1.50. File/Import/PLOT3D/Result File...

The **File/Import/PLOT3D/Result File...** menu item s used to import a PLOT3D result file (extension `.g`, `.x`, `.xyz`, or `.grd`). See *PLOT3D Files* (p. 82) for details.

36.1.51. File/Import/PTC Mechanica Design...

The **File/Import/PTC Mechanica Design...** menu item is used to import a PTC Mechanica Design file (extension `.neu`) which contains analysis, model, and results data (only in binary form). See *PTC Mechanica Design Files* (p. 82) for details.

36.1.52. File/Import/Tecplot...

The **File/Import/Tecplot...** menu item is used to read in a Tecplot binary file. See *Tecplot Files* (p. 83) for details.

36.1.53. File/Import/FLUENT 4 Case File...

The **File/Import/ANSYS FLUENT 4 Case File...** menu item is used to read in a case file created in ANSYS FLUENT 4. See *FLUENT 4 Case Files* (p. 83) for details.

36.1.54. File/Import/PreBFC File...

The **File/Import/PreBFC File...** menu item is used to read in a structured (quadrilateral or hexahedral) mesh that was created using PreBFC. See *PreBFC Files* (p. 83) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.55. File/Import/Partition/Metis...

The **File/Import/Partition/Metis...** menu item is used to partition a mesh and then read it into the parallel version of ANSYS FLUENT. See *Using the Partition Filter* (p. 1622) for an explanation of the difference between this menu item and the **File/Import/Partition/Metis Zone...** menu item.

36.1.56. File/Import/Partition/Metis Zone...

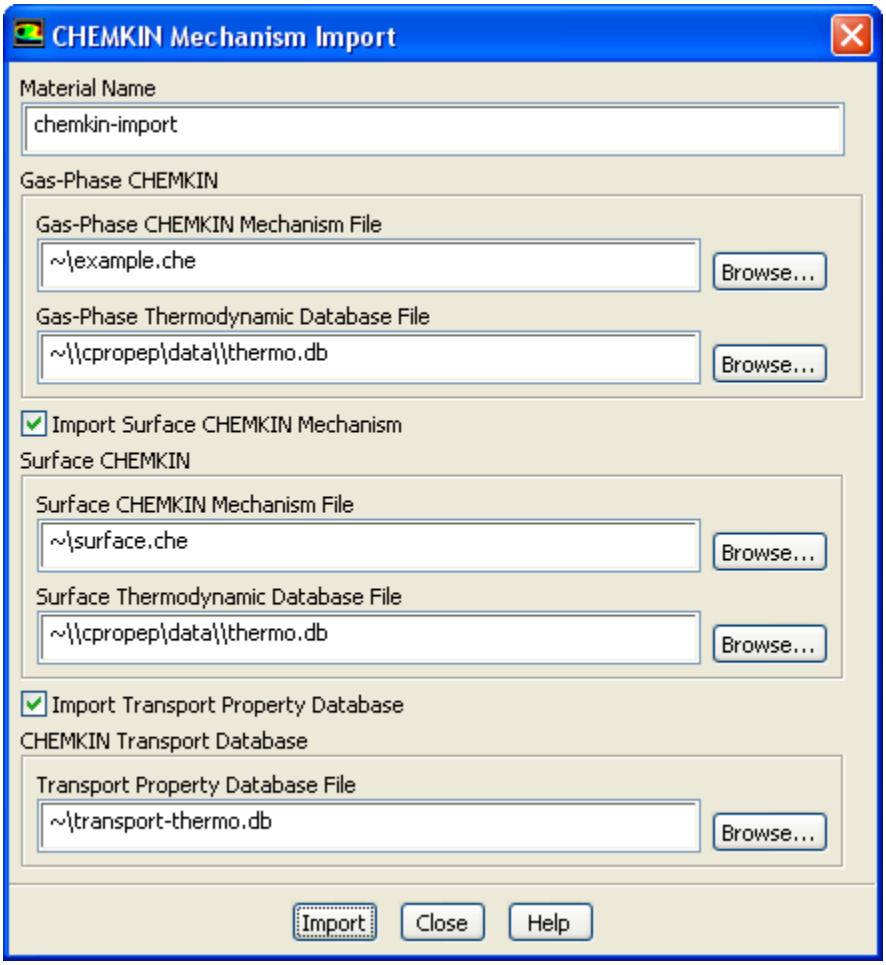
The **File/Import/Partition/Metis Zone...** menu item is used to partition each cell zone in a mesh individually and then read the mesh into the parallel version of ANSYS FLUENT. See *Using the Partition Filter* (p. 1622) for an explanation of the difference between this menu item and the **File/Import/Partition/Metis...** menu item.

36.1.57. File/Import/CHEMKIN Mechanism...

The **File/Import/CHEMKIN Mechanism...** menu item opens the *CHEMKIN Mechanism Import Dialog Box* (p. 2055).

36.1.57.1. CHEMKIN Mechanism Import Dialog Box

The **CHEMKIN Mechanism Import** dialog box is used to import a CHEMKIN-format chemical mechanism file into ANSYS FLUENT. See *Importing a Volumetric Kinetic Mechanism in CHEMKIN Format* (p. 827) for details.



Controls

Material Name
specifies a name for the material.

Gas-Phase CHEMKIN
contains inputs for importing the gas-phase CHEMKIN mechanism.

Gas-Phase CHEMKIN Mechanism File
specifies the name of the gas-phase CHEMKIN-format file. If the file is not in the current working folder, include the full path to the folder where it is located.

Gas-Phase Thermodynamic Database File
specifies the location of the gas-phase thermodynamic database file.

Import Surface CHEMKIN Mechanism
enables the importing of the surface CHEMKIN mechanism.

Surface CHEMKIN
contains inputs for importing the surface CHEMKIN mechanism. This option is available only when the **Import Surface CHEMKIN Mechanism** option is enabled.

Surface CHEMKIN Mechanism File
specifies the name of the surface CHEMKIN-format file.

Surface Thermodynamic Database File
specifies the location of the surface thermodynamic database file.

Import Transport Property Database
enables the importing of the CHEMKIN transport database.

CHEMKIN Transport Database
contains the input for importing the **Transport Property Database File**. This option is available only when the **Import Transport Property Database** option is enabled.

Transport Property Database File
specifies the name of the transport property database file.

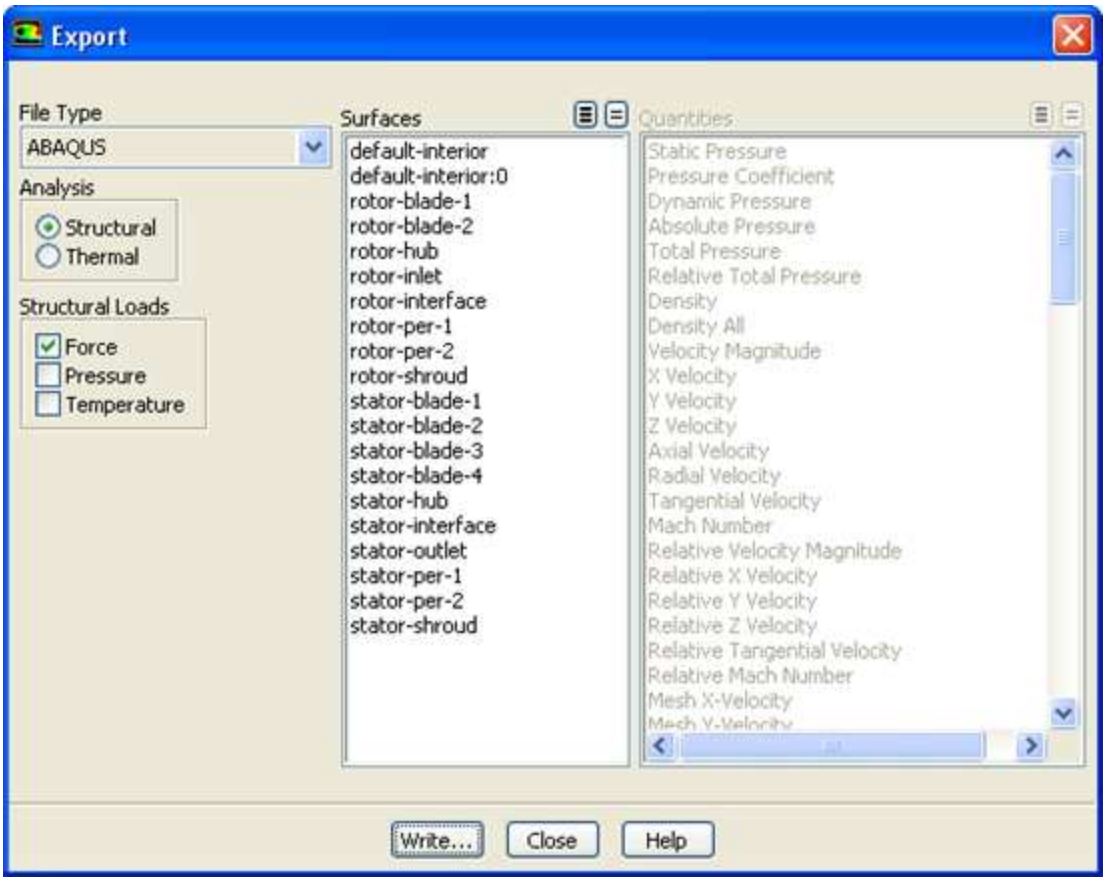
36.1.58. File/Export/Solution Data..

The **File/Export/Solution Data...** menu item opens the *Export Dialog Box* (p. 2056).

36.1.58.1. Export Dialog Box

The **Export** dialog box allows you to write data that can be read by other data visualization and postprocessing tools. See *Exporting Solution Data* (p. 84) for a complete description of the available data formats and how to use the dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

File Type

contains a drop-down list that controls the output file format that will be written using the **Write...** button.

ABAQUS

allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to an ABAQUS file (extension `.inp`).

ASCII

allows you to specify the surface(s), scalars, location from which the values of scalar functions are to be taken, and the delimiter separating the fields, to be exported to an **ASCII** file.

AVS

allows you to specify the scalars you want to write to be exported to an AVS file.

CFD-Post Compatible

allows you to specify the scalars you want to write to be exported to a file compatible to the **CFD-Post** application.

CGNS

allows you to specify the scalars you want to write and the location from which the values of scalar functions are to be taken, to be exported to a **CGNS** file (extension `.cgns`).

Data Explorer

allows you to specify the surface(s) and the scalars you want to write to be exported to a Data Explorer file (extension `.dx`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EnSight Case Gold
allows you to specify the scalars you want to write, the location from which the values of scalar functions are to be taken, and the file format, to be exported to an EnSight file (extension `.geo`, `.vel`, `.scl1`, or `.encas`).

FAST
allows you to specify the scalars you want to write, to be exported as a grid file (Plot3D format), a velocity file, and a scalar file. This option is available only for a triangular or tetrahedral mesh.

FAST Solution
allows you to export a single file containing density, velocity, and total energy data. This option is available only for a triangular or tetrahedral mesh.

Fieldview Unstructured
allows you to specify the scalars you want to write, to be exported to a

I-deas Universal
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to an I-deas Universal file.

Mechanical APDL Input
allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to a Mechanical APDL **Input** file (extension `.cdb`).

NASTRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a NASTRAN file (extension `.bdf`).

PATRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a PATRAN neutral file (extension `.out`).

RadTherm
allows you to specify the surface(s) for which you want to write data and the method of writing the heat transfer coefficient, to be exported to a PATRAN neutral file (extension `.neu`). This option is available only when the **Energy Equation** is enabled.

Tecplot
allows you to specify the surface(s) and the scalars you want to write, to be exported to a Tecplot file.

Surfaces
specifies the surfaces for which data is to be written for an **ABAQUS**, **ASCII**, Data Explorer, I-deas Universal, **Mechanical APDL Input**, NASTRAN, PATRAN, **RadTherm**, or Tecplot file.

Quantities
specifies valid functions for output. The attributes of the list are modified based on the active file type. The list may be a single-selection or a multiple-selection list or it may be disabled, depending on the selected **File Type**.

Structural Loads
contains optional structural loads that can be written to **ABAQUS**, I-deas Universal, **Mechanical APDL Input**, NASTRAN, and PATRAN files. This option is available only when **Structural** analysis is selected.

Force
enables force to be written as a load for a structural analysis.

Pressure
enables pressure to be written as a load for a structural analysis.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature
enables temperature to be written as a load for a structural analysis. This option is available only when the **Energy Equation** is enabled.

Thermal Loads
contains optional thermal loads that can be written to **ABAQUS**, I-deas Universal, **Mechanical APDL Input**, NASTRAN, and PATRAN files. This option is available only when **Thermal** analysis is selected.

Temperature
enables temperature to be written as a load for a thermal analysis.

Heat Flux
enables heat flux to be written as a load for a thermal analysis.

Heat Trans Coeff
enables heat transfer coefficient to be written as a load for a thermal analysis.

Location (for ASCII, CGNS, and EnSight Case Gold formats)
specifies the location from which the values of scalar functions are to be taken.

Node
specifies that data values at the node points are to be exported.

Cell Center
specifies that data values from the cell centers are to be exported.

Analysis (for ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN formats
specifies the finite element analysis intended.

Structural
specifies structural analysis and allows you to select the **Structural Loads** to be written.

Thermal
specifies thermal analysis and allows you to select the **Thermal Loads** to be written.

Delimiter (for ASCIIformat only)
specifies the delimiter used to separate the data fields.

Comma
specifies comma as the delimiter separating the data fields.

Space
specifies space as the delimiter separating the data fields.

Transient (for EnSight Case Gold only)
contains options for exporting transient data.

Transient
enables export of transient data.

Separate Files for Each Timestep
enables writing separate files for each timestep.

Append Frequency
specifies the frequency for appending the data during the solution process.

Important

The **Append Frequency** option is replaced by the **Write Frequency** option when **Separate Files for Each Timestep** is enabled. You can specify the frequency for writing the separate files.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Name
specifies the root name for the files to be saved.

Format (for EnSight Case Goldand CFD-Post Compatible)
specifies the file format.

Binary
specifies the file format as binary.

ASCII
specifies the file format as ASCII.

Heat Transfer Coefficient (for RadTherm format only)
specifies the basis for the heat transfer coefficient exported.

Flux Based
specifies the flux based method for writing the heat transfer coefficient.

Wall Function
specifies the wall function based method for writing the heat transfer coefficient.

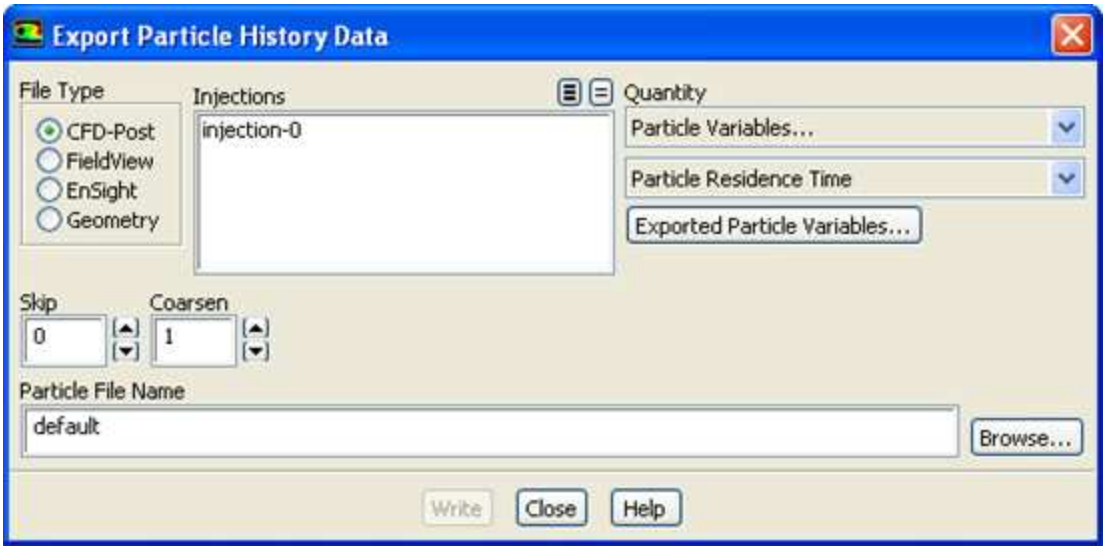
Write...
opens *The **Select File** Dialog Box* (p. 33) for writing the specified function(s) in the specified format.

36.1.59. File/Export/Particle History Data..

The **File/Export/Particle History Data...** menu item opens the *Export Particle History Data Dialog Box* (p. 2060).

36.1.59.1. Export Particle History Data Dialog Box

The **Export Particle History Data** dialog box allows you to export particle history data as your solution progresses. See *Exporting Steady-State Particle History Data* (p. 97) for details.



Controls

Type
specifies the type of the file you want to write.

CFD-Post
allows you to write the file in CFD-Post particle tracks format, which can be read in **CFD-Post**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FieldView
allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

EnSight
allows you to write the file in **EnSight** format.

Geometry
allows you to write the file in `.ibl` format, which can be read in GAMBIT (not available when **Unsteady Particle Tracking** is enabled under the **Define/Models/Discrete Phase...** menu option).

Injections
allows you to select the required injection from the list of predefined injections.

Quantity
contains the list of variables for which you can export the particle data.

Skip
allows you to "thin" or "sample" the number of particles that are exported.

Coarsen
reduces the exported file size by reducing the number of points that are written for a given trajectory. This is only valid for steady-state cases.

Particle File Name
allows you to specify the file name/directory for the exported data, using the **Browse...** button.

Encas File Name
is the file name you will specify if you selected **EnSight** under **Type**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** menu option.

Number of Particle Time Steps
appears when you select **EnSight** under **Type**. The specified number is the number of particle time steps that are saved to the `.encas` file.

36.1.60. File/Export/During Calculation/Solution Data..

The **File/Export/During Calculation/Solution Data...** menu item opens the *Automatic Export Dialog Box* (p. 1946) (transient cases only).

36.1.61. File/Export/During Calculation/Particle History Data..

The **File/Export/During Calculation/Particle History Data...** menu item opens the *Automatic Particle History Data Export Dialog Box* (p. 1949) (transient cases only).

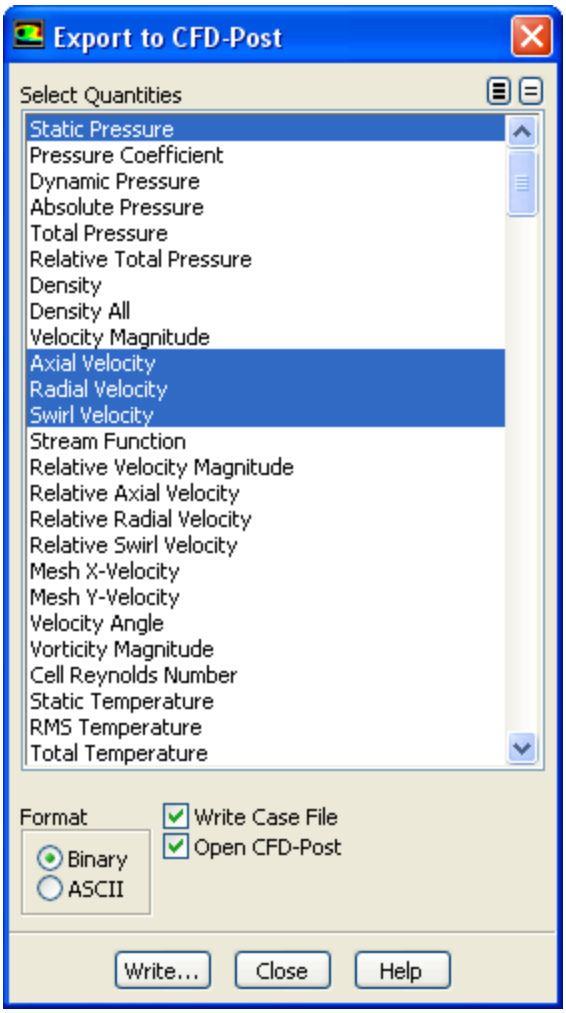
36.1.62. File/Export to CFD-Post...

The **File/Export to CFD-Post...** menu item opens the *Export to CFD-Post Dialog Box* (p. 2061).

36.1.62.1. Export to CFD-Post Dialog Box

The **Export to CFD-Post** dialog box allows you to select the quantities that you would like to export to CFD-Post. See *Exporting to ANSYS CFD-Post* (p. 104) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Select Quantities
contains a selectable list where you can choose the quantities to export.

Format
allows you to save your file in **Binary** or **ASCII** format.

Write Case File
will write the selected quantities to a file.

Open CFD-Post
will open a CFD-Post and the case and `.cdat` files will be loaded. CFD-Post will display the results.

Write...
opens the **Select File** dialog box, where you will save your file.

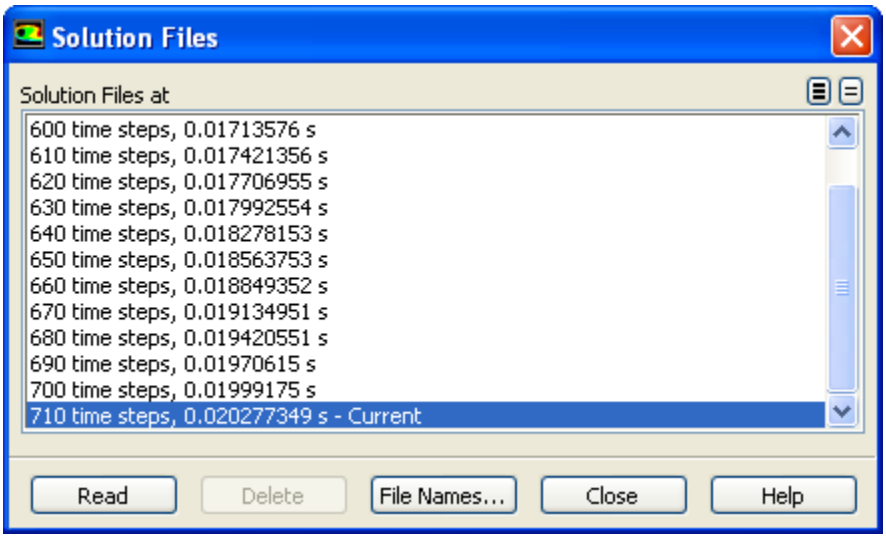
36.1.63. File/Solution Files...

The **File/Solution Files...** menu item opens the *Solution Files Dialog Box* (p. 2062).

36.1.63.1. Solution Files Dialog Box

The **Solution Files** dialog box allows you to manage the files that were created through the *Autosave Dialog Box* (p. 1943). See *Managing Solution Files* (p. 106) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Solution Files at
contains a selectable list where you can choose the files to read or delete.

Read
makes the selected file the current file. Note that if more than one file is selected, the **Read** button is disabled. When an earlier solution is made current, the solution files that were generated for a later iteration/time step will be removed from this list when the calculation continues.

Delete
removes the selected solution files.

File Names...
allows you to obtain information about the solution files and the path of the associated files.

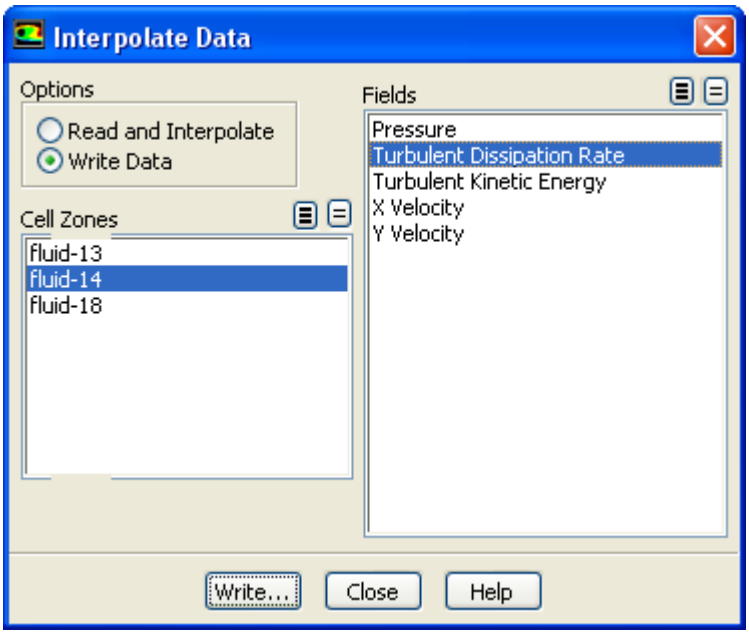
36.1.64. File/Interpolate...

The **File/Interpolate...** menu item opens the *Interpolate Data Dialog Box* (p. 2063).

36.1.64.1. Interpolate Data Dialog Box

The **Interpolate Data** dialog box allows you to interpolate solution data from one mesh to another. See *Mesh-to-Mesh Solution Interpolation* (p. 106) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the interpolation options.

Read and Interpolate

allows you to read and interpolate solution data onto the current mesh.

Write Data

allows you to write an interpolation file for the solution data to be interpolated onto another mesh.

Cell Zones

is a list of cell zones that can be selected.

Fields

is a list of all available data fields that can be selected. This list is available only when **Write Data** is selected under **Options**.

Read...

opens *The **Select File** Dialog Box* (p. 33), in which you can specify the file to be read. This button is available only when the **Read and Interpolate** option is selected.

Write...

opens *The **Select File** Dialog Box* (p. 33), in which you can specify a name for the file to be saved and then save the file. This button replaces the **Read...** button when the **Write Data** option is selected.

36.1.65. File/EM Mapping/HFSS...

The **File/EM Mapping/HFSS...** menu item opens the *HFSS Mapping Dialog Box* (p. 2065).

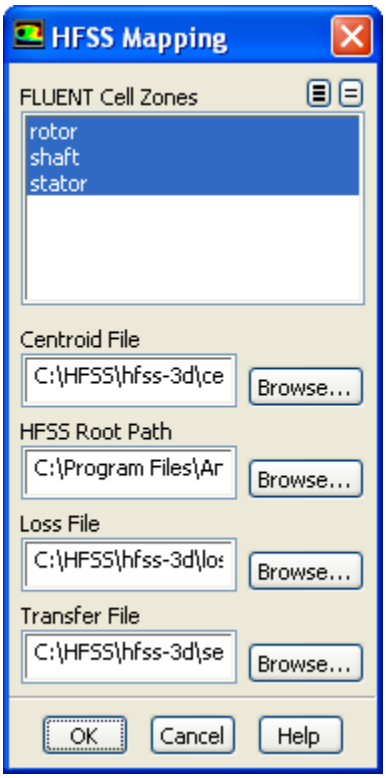
Important

EM Mapping is only available in serial ANSYS FLUENT. It is not available in parallel ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.65.1. HFSS Mapping Dialog Box

The **HFSS Mapping** dialog box allows you to map the volumetric loss results from an Ansoft HFSS electromagnetic simulation onto the centroids of solid cells in the FLUENT mesh as a heat source. See *Mapping Electromagnetic Losses* (p. 109) for details.



Controls

FLUENT Cell Zones
contains a list of solid cell zones from the FLUENT mesh, onto which the loss information can be mapped.

Centroid File
specifies the name of a `.dat` file that FLUENT generates during the mapping process, which provides cell centroid data to HFSS. If you do not want to save the file in the current working folder, include the full path to the appropriate folder.

HFSS Root Path
specifies the root path for the HRSS application.

Loss File
specifies the name of a `.xml` file that HFSS generates during the mapping process, which provides the volumetric loss data to FLUENT. If you do not want to save the file in the current working folder, include the full path to the appropriate folder.

Transfer File
specifies the name of the transfer file you exported from HFSS in anticipation of the mapping process. If the file was not exported to the current working folder, include the full path to the appropriate folder.

36.1.66. File/EM Mapping/Maxwell...

The **File/EM Mapping/Maxwell...** menu item opens the *Maxwell Mapping Dialog Box* (p. 2066).

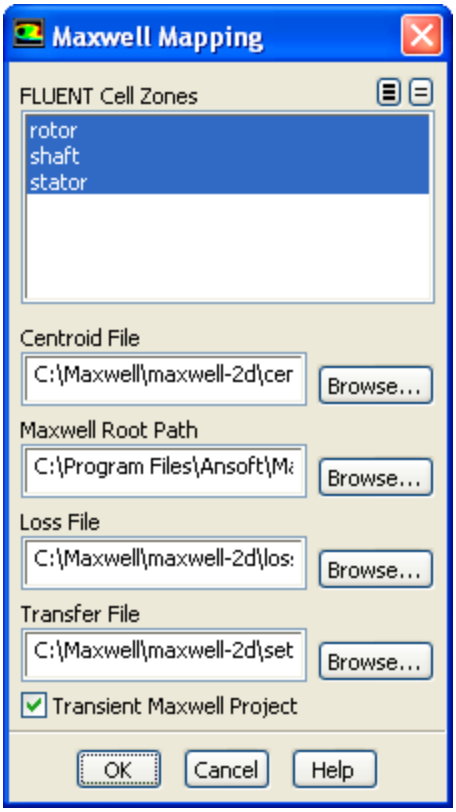
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

EM Mapping is only available in serial ANSYS FLUENT. It is not available in parallel ANSYS FLUENT.

36.1.66.1. Maxwell Mapping Dialog Box

The **Maxwell Mapping** dialog box allows you to map the volumetric loss results from an Ansoft Maxwell electromagnetic simulation onto the centroids of solid cells in the FLUENT mesh as a heat source. See *Mapping Electromagnetic Losses* (p. 109) for details.



Controls

FLUENT Cell Zones
contains a list of solid cell zones from the FLUENT mesh, onto which the loss information can be mapped.

Centroid File
specifies the name of a `.dat` file that FLUENT generates during the mapping process, which provides cell centroid data to Maxwell. If you do not want to save the file in the current working folder, include the full path to the appropriate folder.

Maxwell Root Path
specifies the root path for the Maxwell application.

Loss File
specifies the name of a `.xml` file that Maxwell generates during the mapping process, which provides the volumetric loss data to FLUENT. If you do not want to save the file in the current working folder, include the full path to the appropriate folder.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transfer File

specifies the name of the transfer file you exported from Maxwell in anticipation of the mapping process. If the file was not exported to the current working folder, include the full path to the appropriate folder.

Transient Maxwell Project

specifies whether or not the Maxwell simulation run in anticipation of the mapping process was transient.

36.1.67. File/EM Mapping/Q3D...

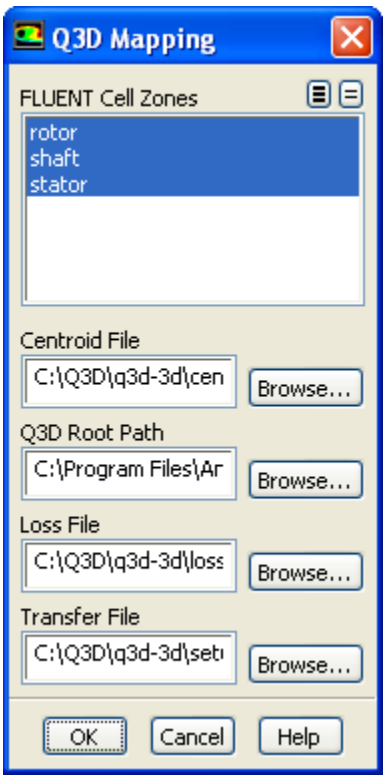
The **File/EM Mapping/Q3D...** menu item opens the *Q3D Mapping Dialog Box* (p. 2067).

Important

EM Mapping is only available in serial ANSYS FLUENT. It is not available in parallel ANSYS FLUENT.

36.1.67.1. Q3D Mapping Dialog Box

The **Q3D Mapping** dialog box allows you to map the volumetric loss results from an Ansoft Q3D Extractor electromagnetic simulation onto the centroids of solid cells in the FLUENT mesh as a heat source. See *Mapping Electromagnetic Losses* (p. 109) for details.



Controls

FLUENT Cell Zones

contains a list of solid cell zones from the FLUENT mesh, onto which the loss information can be mapped.

Centroid File

specifies the name of a `.dat` file that FLUENT generates during the mapping process, which provides cell centroid data to Q3D Extractor. If you do not want to save the file in the current working folder, include the full path to the appropriate folder.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Q3D Root Path

specifies the root path for the Q3D Extractor application.

Loss File

specifies the name of a `.xml` file that Q3D Extractor generates during the mapping process, which provides the volumetric loss data to FLUENT. If you do not want to save the file in the current working folder, include the full path to the appropriate folder.

Transfer File

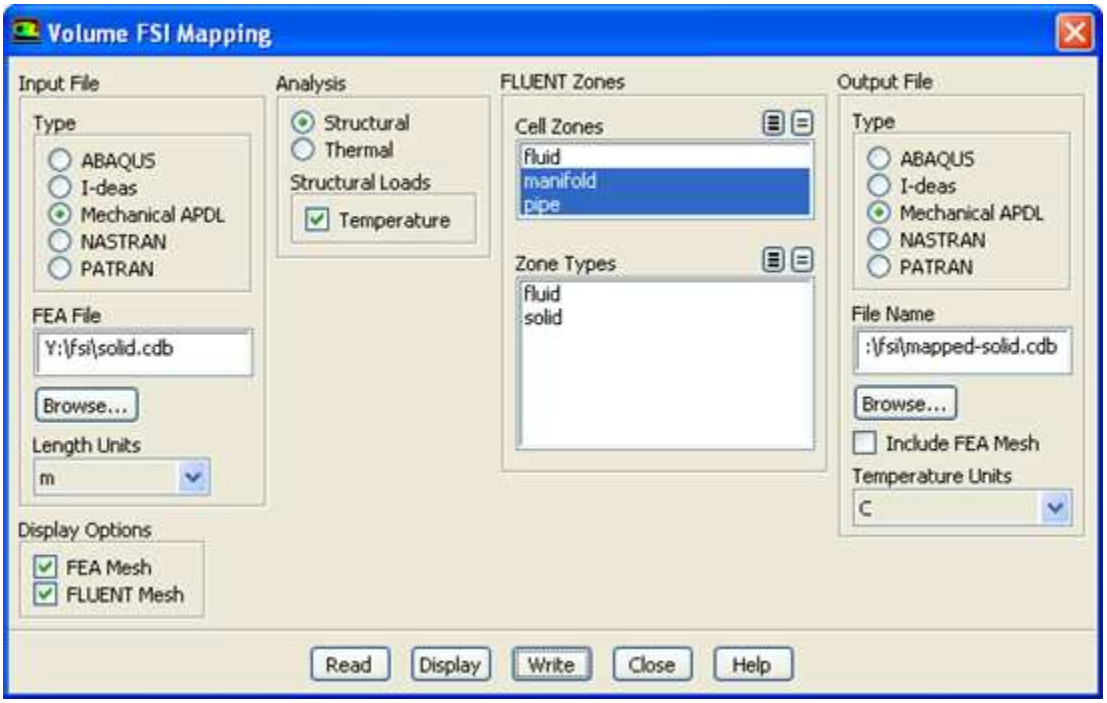
specifies the name of the transfer file you exported from Q3D Extractor in anticipation of the mapping process. If the file was not exported to the current working folder, include the full path to the appropriate folder.

36.1.68. File/FSI Mapping/Volume...

The **File/FSI Mapping/Volume...** menu item opens the *Volume FSI Mapping Dialog Box* (p. 2068).

36.1.68.1. Volume FSI Mapping Dialog Box

The **Volume FSI Mapping** dialog box allows you to map cell data for a given geometry from an ANSYS FLUENT file onto a file with a different mesh and format. See *Mapping Data for Fluid-Structure Interaction (FSI) Applications* (p. 113) for a complete description of how to use the dialog box.



Controls

Input File

contains parameters related to the input file.

Type

contains the options for the format of the input mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the input mesh file (extension `.inp`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I-deas
specifies that an I-deas file will be used as the input mesh file (extension `.unv`).

Mechanical APDL
specifies that a Mechanical APDL file will be used as the input mesh file (extension `.cdb`).

NASTRAN
specifies that a NASTRAN file will be used as the input mesh file (extension `.bdf`).

PATRAN
specifies that a PATRAN file will be used as the input mesh file (extension `.out`).

FEA File
specifies the name of the input mesh file (see the **Input File Type** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...
opens the *The **Select File** Dialog Box* (p. 33), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Length Units
specifies the unit of length used in the input file.

Display Options
allows you to choose either the **FEA Mesh** or the **ANSYS FLUENT Mesh** (or both).

Analysis
contains the options for the kind of data to be mapped.

Structural
specifies that data fields relevant for a structural analysis will be mapped.

Thermal
specifies that data fields relevant for a thermal analysis will be mapped.

Structural Loads
consists of a list of available loads, including **Force**, **Pressure**, and **Temperature**.

FLUENT Zones
contains a list of available cell zones from the current ANSYS FLUENT file from which cell data will be mapped.

Output File
contains the options for the format of the file that will be created from the input mesh file and the mapped data.

Type
contains the options for the format of the output mesh file.

ABAQUS
specifies that an ABAQUS file will be used as the output mesh file (extension `.inp`).

I-deas
specifies that an I-deas file will be used as the output mesh file (extension `.unv`).

Mechanical APDL
specifies that a Mechanical APDL file will be used as the output mesh file (extension `.cdb`).

NASTRAN
specifies that a NASTRAN file will be used as the output mesh file (extension `.bdf`).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PATRAN
specifies that a PATRAN file will be used as the output mesh file (extension `.out`).

File Name
specifies the name of the input mesh file (see the **Output File** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...
opens the *The **Select File** Dialog Box* (p. 33), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Include FEA Mesh
includes additional FEA information like node/element information in the exported output file.

Temperature Units
specifies the unit of temperature when mapping temperature for a structural analysis or any variable for a thermal analysis.

Read
reads the input file into memory.

Display
displays the selected mesh in the graphics window.

Write
writes an output file in which the selected ANSYS FLUENT cell data is mapped onto the input mesh file.

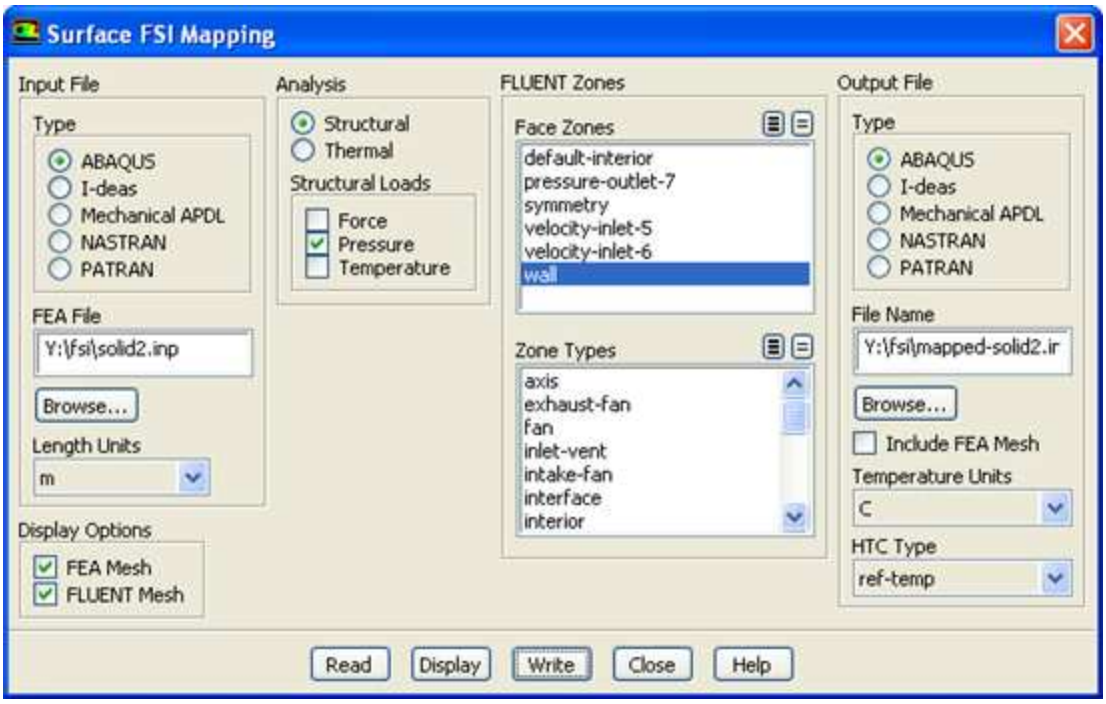
36.1.69. File/FSI Mapping/Surface...

The **File/FSI Mapping/Surface...** menu item opens the *Surface FSI Mapping Dialog Box* (p. 2070).

36.1.69.1. Surface FSI Mapping Dialog Box

The **Surface FSI Mapping** dialog box allows you to map face data for a given geometry from an ANSYS FLUENT file to a file with a different mesh and format. See *Mapping Data for Fluid-Structure Interaction (FSI) Applications* (p. 113) for a complete description of how to use the dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Input File

contains parameters related to the input file.

Type

contains the options for the format of the input mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the input mesh file (extension `.inp`).

I-deas

specifies that an I-deas file will be used as the input mesh file (extension `.unv`).

Mechanical APDL

specifies that a Mechanical APDL file will be used as the input mesh file (extension `.cdb`).

NASTRAN

specifies that a NASTRAN file will be used as the input mesh file (extension `.bdf`).

PATRAN

specifies that a PATRAN file will be used as the input mesh file (extension `.out`).

FEA File

specifies the name of the input mesh file (see the **Input File Type** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...

opens the *The **Select File** Dialog Box* (p. 33), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Length Units

specifies the unit of length used in the input file.

Display Options

allows you to choose either the **FEA Mesh** or the **ANSYS FLUENT Mesh** (or both).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis
contains the options for the kind of data to be mapped.

Structural
specifies that data fields relevant for a structural analysis will be mapped.

Thermal
specifies that data fields relevant for a thermal analysis will be mapped.

Structural Loads
consists of a list of available loads, including **Force**, **Pressure**, and **Temperature**.

FLUENT Zones
contains a list of available face zones from the current ANSYS FLUENT file from which cell data will be mapped.

Output File
contains the options for the format of the file that will be created from the input mesh file and the mapped data.

Type
contains the options for the format of the output mesh file.

ABAQUS
specifies that an ABAQUS file will be used as the output mesh file (extension `.inp`).

I-deas
specifies that an I-deas file will be used as the output mesh file (extension `.unv`).

Mechanical APDL
specifies that a Mechanical APDL file will be used as the output mesh file (extension `.cdb`).

NASTRAN
specifies that a NASTRAN file will be used as the output mesh file (extension `.bdf`).

PATRAN
specifies that a PATRAN file will be used as the output mesh file (extension `.out`).

File Name
specifies the name of the input mesh file (see the **Output File** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...
opens the *The **Select File** Dialog Box* (p. 33), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Include FEA Mesh
includes additional FEA information like node/element information in the exported output file.

Temperature Units
specifies the unit of temperature when mapping temperature for a structural analysis or any variable for a thermal analysis.

HTC Type
specifies the means of calculating the heat transfer coefficient.

ref-temp
uses a temperature equal to the reference temperature to calculate the heat transfer coefficient.

cell-temp
uses a temperature equal to the temperature of the cell adjacent to the face to calculate the heat transfer coefficient.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall-func-htc

calculates h_{eff} using *Equation 33–55* (p. 1578). Note that this option has the same definition as the field variable **Wall Func. Heat Tran. Coef.**, as described in *Alphabetical Listing of Field Variables and Their Definitions* (p. 1547).

Read

reads the input file into memory.

Display

displays the selected mesh in the graphics window.

Write

writes an output file in which the ANSYS FLUENT data has been mapped to the mesh of the input file.

36.1.70. File/Save Picture...

The **File/Save Picture...** menu item opens the *Save Picture Dialog Box* (p. 1988).

36.1.71. File/Data File Quantities...

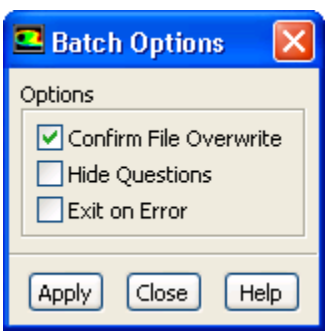
The **File/Data File Quantities...** menu item opens the *Data File Quantities Dialog Box* (p. 1945).

36.1.72. File/Batch Options...

The **File/Batch Options...** menu item opens the *Batch Options Dialog Box* (p. 2073).

36.1.72.1. Batch Options Dialog Box

The **Batch Options** dialog box allows you to select options to suppress interactive dialog boxes from ANSYS FLUENT while running a case in batch mode. See *Batch Execution Options* (p. 16) for details.



Controls

Confirm File Overwrite

determines whether ANSYS FLUENT confirms a file overwrite. This option is enabled by default.

Hide Questions

allows you to hide **Question** dialog boxes. This option is disabled by default.

Exit on Error

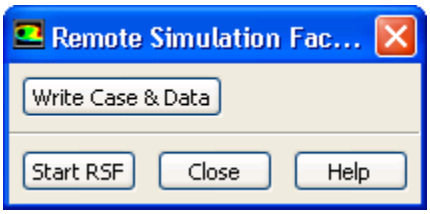
allows you to automatically exit from batch mode when an error occurs. This option is disabled by default.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.73. File/RSF...

ANSYS FLUENT's Remote Simulation Facility (RSF) is a highly secure mechanism for executing large, complex simulations at a remote data center.

The **File/RSF...** menu item opens the **Remote Simulation Facility (RSF)** dialog box.



Controls

Write Case & Data
allows you to write case and data files for the current session. You must save the case and data files in order to use them on the RSF. This option is available only after the case and data files are already loaded into ANSYS FLUENT.

Start RSF
opens your web browser and loads the page for the ANSYS FLUENT Remote Simulation Facility.

36.1.74. File/Exit

The **File/Exit** menu item is used to exit from the current solver session.

36.2. Mesh Menu

For additional information, please see the following sections:

36.2.1. Mesh/Check
36.2.2. Mesh/Info/Quality
36.2.3. Mesh/Info/Size
36.2.4. Mesh/Info/Memory Usage
36.2.5. Mesh/Info/Zones
36.2.6. Mesh/Info/Partitions
36.2.7. Mesh/Polyhedra/Convert Domain
36.2.8. Mesh/Polyhedra/Convert Skewed Cells...
36.2.9. Mesh/Merge...
36.2.10. Mesh/Separate/Faces...
36.2.11. Mesh/Separate/Cells...
36.2.12. Mesh/Fuse...
36.2.13. Mesh/Zone/Append Case File...
36.2.14. Mesh/Zone/Append Case & Data Files...
36.2.15. Mesh/Zone/Replace...
36.2.16. Mesh/Zone/Delete...
36.2.17. Mesh/Zone/Deactivate...
36.2.18. Mesh/Zone/Activate...
36.2.19. Mesh/Replace...
36.2.20. Mesh/Reorder/Domain
36.2.21. Mesh/Reorder/Zones
36.2.22. Mesh/Reorder/Print Bandwidth

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.23. Mesh/Scale...
36.2.24. Mesh/Translate...
36.2.25. Mesh/Rotate...
36.2.26. Mesh/Smooth/Swap...

36.2.1. Mesh/Check

The **Mesh/Check** menu item is used to verify the validity of the mesh. See *Checking the Mesh* (p. 172) for details.

36.2.2. Mesh/Info/Quality

The **Mesh/Info/Quality** menu item is used to display information about the quality of the mesh in the console, including the minimum orthogonal quality and the maximum aspect ratio. See *Mesh Quality* (p. 144) for details.

36.2.3. Mesh/Info/Size

The **Mesh/Info/Size** menu item is used to print out the number of nodes, faces, cells, and partitions in the mesh. See *Mesh Size* (p. 175) for details.

36.2.4. Mesh/Info/Memory Usage

The **Mesh/Info/Memory Usage** menu item is used to check the amount of memory used and allocated in the present analysis.

This feature reports the following information: the number of nodes, faces, cells, edges, and object pointers (generic pointers for various mesh and graphics utilities) that are used and allocated; the amount of array memory (scratch memory used for surfaces) used and allocated; and the amount of memory used by the solver process.

The memory information will be different for UNIX and Windows systems. See *Memory Usage* (p. 176) for details.

36.2.5. Mesh/Info/Zones

The **Mesh/Info/Zones** menu item is used to print the total number of nodes for each face and cell zone, the number of faces or cells, the cell (and, in 3D, face) type (triangular, quadrilateral, etc.), the boundary condition type, and the zone ID. See *Mesh Zone Information* (p. 177) for details.

36.2.6. Mesh/Info/Partitions

The **Mesh/Info/Partitions** menu item is used to print out mesh partition statistics in the console window. See *Interpreting Partition Statistics* (p. 1622) for details.

Important

This report is the same as the one generated using the **Print Partitions** button in the *Partitioning and Load Balancing Dialog Box* (p. 2161).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.7. Mesh/Polyhedra/Convert Domain

The **Mesh/Polyhedra/Convert Domain** menu item is used to convert all 3D meshes (except pure hex meshes) to polyhedral cells. See *Converting the Domain to a Polyhedra* (p. 178) for details.

Important

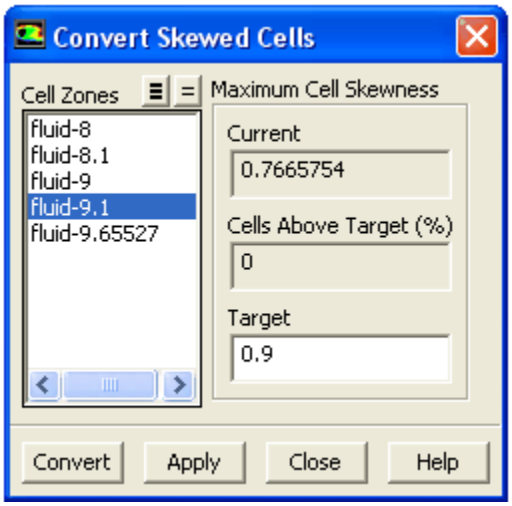
Conversion of a mesh to polyhedra only applies to 3D meshes that contain tetrahedral and/or wedge/prism cells. Hexahedral cells remain unchanged during conversion

36.2.8. Mesh/Polyhedra/Convert Skewed Cells...

The **Mesh/Polyhedra/Convert Skewed Cells...** menu item opens the *Convert Skewed Cells Dialog Box* (p. 2076).

36.2.8.1. Convert Skewed Cells Dialog Box

The **Convert Skewed Cells** dialog box allows you to convert part of your domain to polyhedral cells. See *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195) for details. For information about converting skewed cells see *Using the Convert Skewed Cells Dialog Box* (p. 182).



Controls

Cell Zones
displays the zones that can be selected for cell conversion.

Maximum Cell Skewness
displays the cells skewness parameters of the current mesh.

Current
displays the current maximum cell skewness of the mesh.

Cells Above Target (%)
displays the percentage of cells which are above the **Target** skewness.

Important

The **Cells Above Target (%)** should be only a couple of percentage points, else the conversion will be ineffective due to the high face count.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target
allows you to specify the maximum allowable cell skewness.

Important

To update the **Cells Above Target** (%) Press `<Enter>` after entering **Target** value.

Convert
converts the selected zones to polyhedral cells.

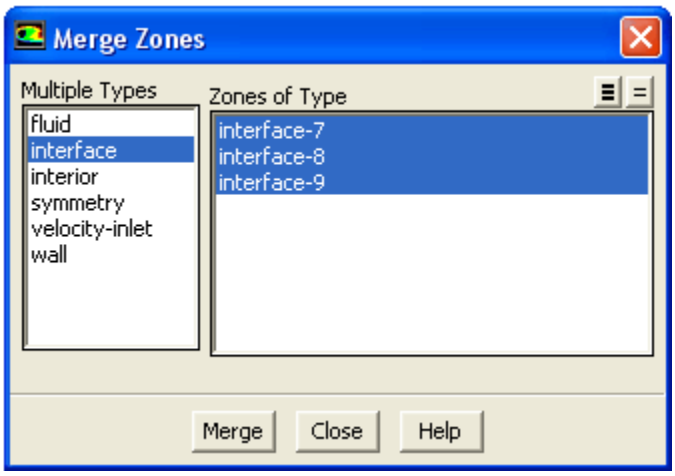
Apply
saves the value entered for **Target**.

36.2.9. Mesh/Merge...

The **Mesh/Merge...** menu item opens the *Merge Zones Dialog Box* (p. 2077).

36.2.9.1. Merge Zones Dialog Box

The **Merge Zones** dialog box allows you to merge multiple zones of the same type into a single zone. See *Merging Zones* (p. 184) for details.



Controls

Multiple Types
contains a selectable list of the zone types for which you can merge multiple zones. You select a type and the corresponding zones of that type will be displayed in the **Zones of Type** list.

Zones of Type
contains a list from which you can select two or more zones to be merged into a single zone. The zones are of the type selected in the **Multiple Types** list.

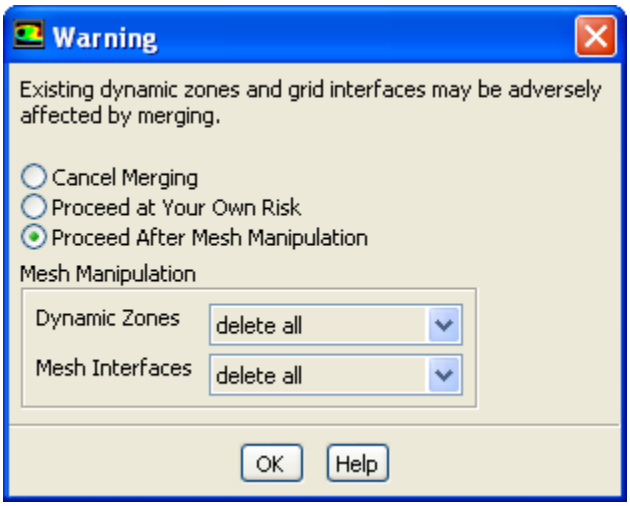
Merge
merges the selected zones into a single zone. If your case file has dynamic zones or mesh interfaces, the **Warning** dialog box will open and require your input prior to the merge.

36.2.9.2. Warning Dialog Box

The **Warning** dialog box allows you to specify whether existing dynamic zones and/or mesh interfaces are deleted when you are using the **Merge Zones** dialog box to merge multiple zones of the same type into a

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

single zone (see *Merge Zones Dialog Box* (p. 2077) for details). Note that dynamic zones and mesh interfaces that exist at the time of the merge may be adversely affected by the merge.



Controls

Cancel Merging
returns you to the **Mesh Zones** dialog box when you click **OK**, without initiating the merge.

Proceed at Your Own Risk
allows the merge to proceed when you click **OK**, without deleting the existing dynamic zones and mesh interfaces beforehand.

Proceed After Mesh Manipulation
deletes the existing dynamic zones and/or mesh interfaces (based on the settings in the **Mesh Manipulation** group box) and then merges the zones when you click **OK**.

Mesh Manipulation
allows you to specify whether the existing dynamic zones and/or mesh interfaces are deleted prior to the initiation of the merge. This group box is only available when **Proceed After Mesh Manipulation** is selected.

Dynamic Zones
specifies whether all existing dynamic zones are deleted prior to the merge.

Mesh Interfaces
specifies whether all existing mesh interfaces are deleted prior to the merge.

Note that selecting **keep all** from both drop-down lists in the **Mesh Manipulation** group box is the equivalent of selecting **Proceed at Your Own Risk**.

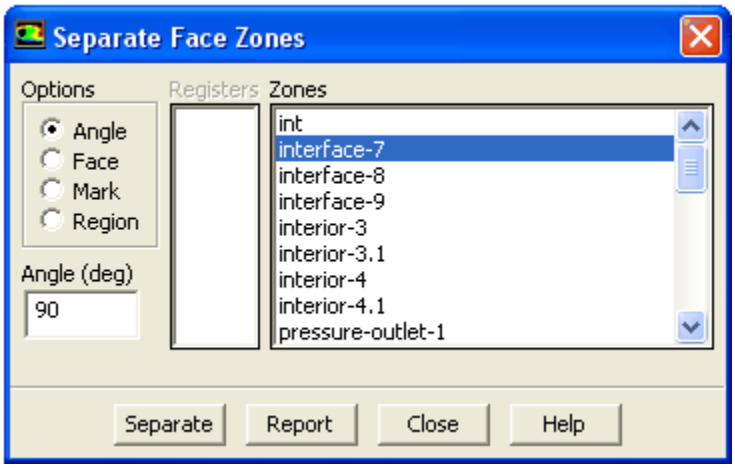
36.2.10. Mesh/Separate/Faces...

The **Mesh/Separate/Faces...** menu item opens the *Separate Face Zones Dialog Box* (p. 2078).

36.2.10.1. Separate Face Zones Dialog Box

The **Separate Face Zones** dialog box allows you to separate a single face zone into multiple zones of the same type. See *Separating Zones* (p. 185) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

specifies the method on which the face separation is to be based.

Angle

indicates that the face zone is to be separated based on significant angle (specified in the **Angle** field).

Face

indicates that the face zone is to be separated by putting each face in the zone into its own zone.

Mark

indicates that the face zone is to be separated based on the marks stored in adaption registers.

Region

indicates that the face zone is to be separated based on contiguous regions.

Registers

contains a list of defined adaption registers. If you are separating faces by mark, select the adaption register to be used in the **Registers** list. When the separation is performed, all faces of cells that are marked will be placed into a new face zone.

Zones

contains a list of face zones from which you can select the zone to be separated.

Angle

specifies the significant angle to be used when you separate a face zone based on angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle will be placed in different zones when the separation occurs.

Separate

separates the selected face zone based on the specified parameters.

Report

reports what the result of the separation will be without actually separating the face zone.

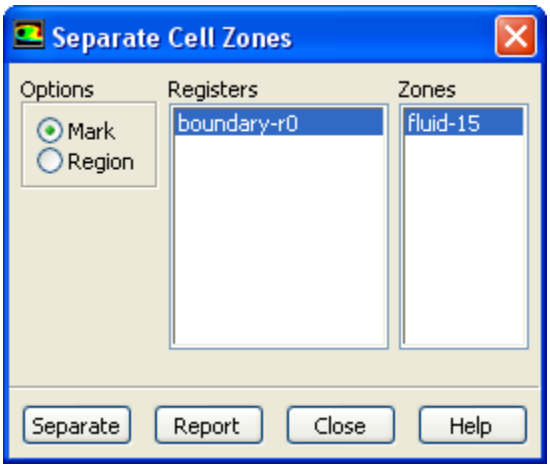
36.2.11. Mesh/Separate/Cells...

The **Mesh/Separate/Cells...** menu item opens the *Separate Cell Zones Dialog Box* (p. 2080).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.11.1. Separate Cell Zones Dialog Box

The **Separate Cell Zones** dialog box allows you to separate a single cell zone into multiple zones of the same type. See *Separating Zones* (p. 185) for details.



Controls

Options
specifies the method on which the cell-zone separation is to be based.

Mark
indicates that the cell zone is to be separated based on the marks stored in adaption registers.

Region
indicates that the cell zone is to be separated into two or more contiguous regions based on an internal boundary within the original zone.

Registers
contains a list of defined adaption registers. If you are separating the cell zone by mark, select the adaption register to be used in the **Registers** list. When the separation is performed, cells that are marked will be placed into a new zone.

Zones
contains a list of cell zones from which you can select the zone to be separated.

Separate
separates the selected cell zone based on the specified parameters.

Report
reports what the result of the separation will be without actually separating the cell zone.

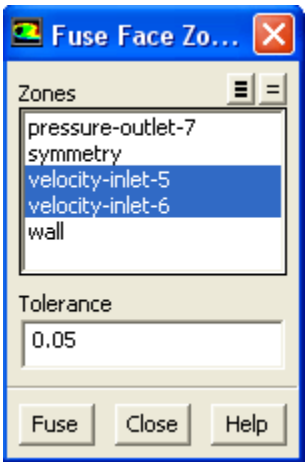
36.2.12. Mesh/Fuse...

The **Mesh/Fuse...** menu item opens the *Fuse Face Zones Dialog Box* (p. 2080).

36.2.12.1. Fuse Face Zones Dialog Box

The **Fuse Face Zones** dialog box allows you to fuse boundaries (i.e., remove duplicate nodes and faces and delete artificial internal boundaries) created by assembling multiple mesh regions. (See *Reading Multiple Mesh/Case/Data Files* (p. 157) for details on importing such meshes.) See *Fusing Face Zones* (p. 190) for information about using this dialog box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zones
contains a list of face zones from which you can select the boundaries to be fused.

Tolerance
is a fraction of the minimum edge length of the face, used to determine whether or not nodes are coincident. If all of the appropriate faces do not get fused using the default **Tolerance**, you should increase it and attempt to fuse the zones again. The **Tolerance** should not exceed 0.5, or you may fuse the wrong nodes.

Fuse
fuses the selected zones. It is enabled only when you have selected a minimum of two zones.

36.2.13. Mesh/Zone/Append Case File...

The **Mesh/Zone/Append Case File...** menu item opens the **Select File** dialog box. This allows you to handle more than one mesh at a time within the same solver settings. See *Reading Multiple Mesh/Case/Data Files* (p. 157) for details.

36.2.14. Mesh/Zone/Append Case & Data Files...

The **Mesh/Zone/Append Case & Data Files...** menu item opens the **Select File** dialog box. This allows you to append data on the mesh. See *Reading Multiple Mesh/Case/Data Files* (p. 157) for details.

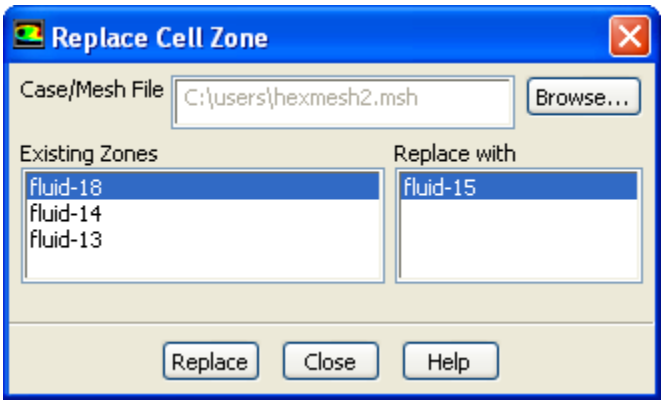
36.2.15. Mesh/Zone/Replace...

The **Mesh/Zone/Replace...** menu item opens the *Replace Cell Zone Dialog Box* (p. 2081)

36.2.15.1. Replace Cell Zone Dialog Box

The **Replace Cell Zone** dialog box allows you to replace a single cell zone or multiple zones. See *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Case/Mesh File
allows you to specify a mesh file from which you want to replace the zone.

Existing Zones
contains a list of cell zones from which you can select the zone to be replaced.

Replace with
contains a list of cell zones from which you can select the zone to replace the zone selected in **Existing Zones** list.

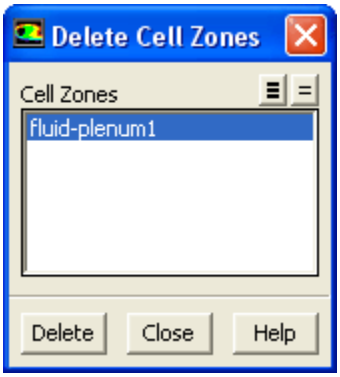
Replace
replaces the selected cell zone.

36.2.16. Mesh/Zone/Delete...

The **Mesh/Zone/Delete...** menu item opens the *Delete Cell Zones Dialog Box* (p. 2082)

36.2.16.1. Delete Cell Zones Dialog Box

The **Delete Cell Zones** dialog box allows you to delete a single cell zone or multiple zones. See *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195) for details.



Controls

Cell Zones
contains a list of cell zones from which you can select the zone to be deleted.

Delete
deletes the selected cell zones.

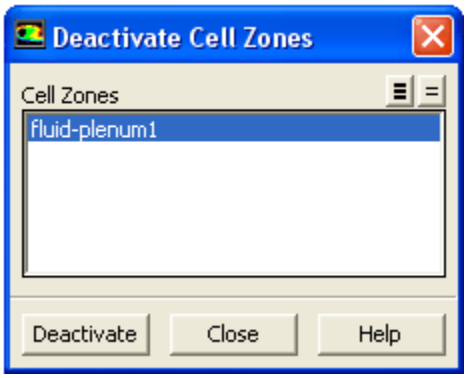
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.17. Mesh/Zone/Deactivate...

The **Mesh/Zone/Deactivate...** menu item opens the *Deactivate Cell Zones Dialog Box* (p. 2083).

36.2.17.1. Deactivate Cell Zones Dialog Box

The **Deactivate Cell Zones** dialog box allows you to deactivate a single cell zone or multiple zones. See *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195) for details.



Controls

Cell Zones
contains a list of cell zones from which you can select the zone to be deactivated.

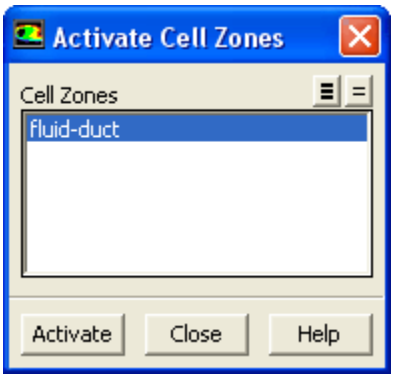
Deactivate
deactivates the selected cell zones.

36.2.18. Mesh/Zone/Activate...

The **Mesh/Zone/Activate...** menu item opens the *Activate Cell Zones Dialog Box* (p. 2083).

36.2.18.1. Activate Cell Zones Dialog Box

The **Activate Cell Zones** dialog box allows you to activate a single cell zone or multiple zones. See *Replacing, Deleting, Deactivating, and Activating Zones* (p. 195) for details.



Controls

Cell Zones
contains a list of cell zones from which you can select the zone to be activated.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Activate
activates the selected cell zones.

36.2.19. Mesh/Replace...

The **Mesh/Replace...** menu item is used to replace the global mesh by reading an ANSYS FLUENT mesh (`.msh`) file. If applicable, data will be interpolated onto the new mesh. See *Replacing the Mesh* (p. 198) for details.

The **Mesh/Replace...** menu item opens the **Select File** dialog box (see *The **Select File** Dialog Box* (p. 33)), which allows you to select the appropriate file to be read.

36.2.20. Mesh/Reorder/Domain

The **Mesh/Reorder/Domain** menu item will reorder the nodes, faces, and cells along zones and in memory in order to increase memory access efficiency. See *Reordering the Domain and Zones* (p. 199) for details.

36.2.21. Mesh/Reorder/Zones

The **Mesh/Reorder/Zones** menu item will reorder the zones first by type and then by ID, for user-interface convenience. See *Reordering the Domain and Zones* (p. 199) for details.

36.2.22. Mesh/Reorder/Print Bandwidth

The **Mesh/Reorder/Print Bandwidth** menu item is used to print the semi-bandwidth and maximum memory distance for each mesh partition. See *Reordering the Domain and Zones* (p. 199) for details.

36.2.23. Mesh/Scale...

The **Mesh/Scale...** menu item opens the *Scale Mesh Dialog Box* (p. 1634).

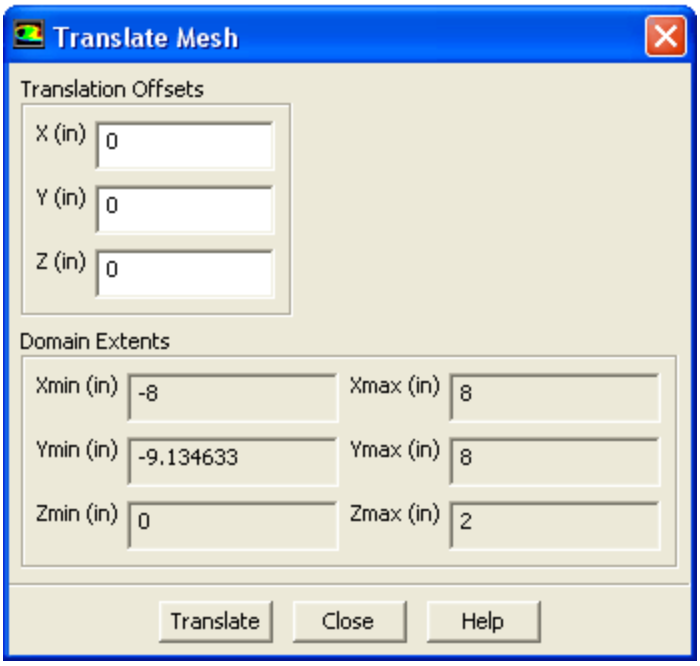
36.2.24. Mesh/Translate...

The **Mesh/Translate...** menu item opens the *Translate Mesh Dialog Box* (p. 2084).

36.2.24.1. Translate Mesh Dialog Box

The **Translate Mesh** dialog box allows you to change the origin of the mesh. See *Translating the Mesh* (p. 202) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Translation Offsets

contains the desired changes in the mesh coordinates (i.e., the desired delta in the axes origin). You can enter a positive or negative real number.

X,Y,Z

defines the deltas in the x, y, and z directions, in the current units of length.

Domain Extents

displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Xmin, Ymin, Zmin

shows the minimum values of Cartesian coordinates in the mesh.

Xmax, Ymax, Zmax

shows the maximum values of Cartesian coordinates in the mesh.

Translate

adds the specified translation offsets to the appropriate Cartesian coordinate of every node in the mesh.

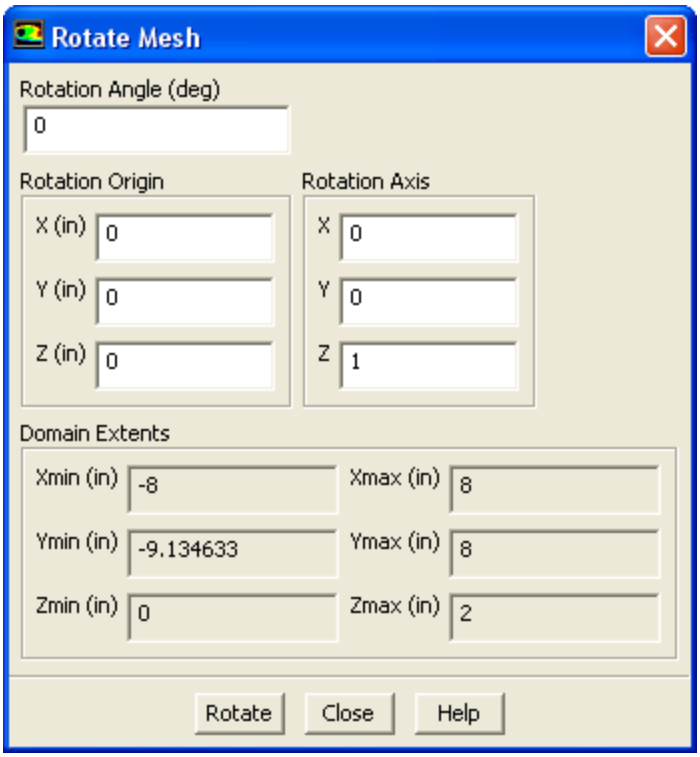
36.2.25. Mesh/Rotate...

The **Mesh/Rotate...** menu item opens the *Rotate Mesh Dialog Box* (p. 2085).

36.2.25.1. Rotate Mesh Dialog Box

The **Rotate Mesh** dialog box allows you to rotate the mesh about the required axis and rotation origin by specifying the angle of rotation. See *Rotating the Mesh* (p. 203) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Rotation Angle
is the angle with which you want to rotate the mesh. You can enter a positive or negative real number.

Rotation Origin
defines the new origin for the mesh rotation.

Rotation Axis
defines the axis about which you want to rotate the mesh.

Domain Extents
displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Rotate
adds the specified rotation parameters to the appropriate Cartesian coordinate of every node in the mesh.

36.2.26. Mesh/Smooth/Swap...

The **Mesh/Smooth/Swap...** menu item opens the *Smooth/Swap Mesh Dialog Box* (p. 2135), which controls smoothing and diagonal swapping. See *Improving the Mesh by Smoothing and Swapping* (p. 1349) for details about its use.

36.3. Define Menu

For additional information, please see the following sections:

36.3.1. Define/General...
36.3.2. Define/Models...
36.3.3. Define/Materials...
36.3.4. Define/Phases...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.5. Define/Cell Zone Conditions...
36.3.6. Define/Boundary Conditions...
36.3.7. Define/Operating Conditions...
36.3.8. Define/Mesh Interfaces...
36.3.9. Define/Dynamic Mesh...
36.3.10. Define/Mesh Morpher/Optimizer...
36.3.11. Define/Mixing Planes...
36.3.12. Define/Turbo Topology...
36.3.13. Define/Injections...
36.3.14. Define/DTRM Rays...
36.3.15. Define/Shell Conduction Walls...
36.3.16. Define/Custom Field Functions...
36.3.17. Define/Parameters...
36.3.18. Define/Profiles...
36.3.19. Define/Units...
36.3.20. Define/User-Defined/Functions/Interpreted...
36.3.21. Define/User-Defined/Functions/Compiled...
36.3.22. Define/User-Defined/Functions/Manage...
36.3.23. Define/User-Defined/Function Hooks...
36.3.24. Define/User-Defined/Execute on Demand...
36.3.25. Define/User-Defined/Scalars...
36.3.26. Define/User-Defined/Memory...
36.3.27. Define/User-Defined/Fan Model...
36.3.28. Define/User-Defined/1D Coupling...

36.3.1. Define/General...

The **Define/General...** menu item opens the *General Task Page* (p. 1631).

36.3.2. Define/Models...

The **Define/Models...** menu item opens the *Models Task Page* (p. 1639).

36.3.3. Define/Materials...

The **Define/Materials...** menu item opens the *Materials Task Page* (p. 1735).

36.3.4. Define/Phases...

The **Define/Phases...** menu item opens the *Phases Task Page* (p. 1783).

36.3.5. Define/Cell Zone Conditions...

The **Define/Cell Zone Conditions...** menu item opens the *Cell Zone Conditions Task Page* (p. 1794).

36.3.6. Define/Boundary Conditions...

The **Define/Boundary Conditions...** menu item opens the *Boundary Conditions Task Page* (p. 1812).

36.3.7. Define/Operating Conditions...

The **Define/Operating Conditions...** menu item opens the *Operating Conditions Dialog Box* (p. 1806).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.8. Define/Mesh Interfaces...

The **Define/Mesh Interfaces...** menu item opens the *Mesh Interfaces Task Page* (p. 1875).

36.3.9. Define/Dynamic Mesh...

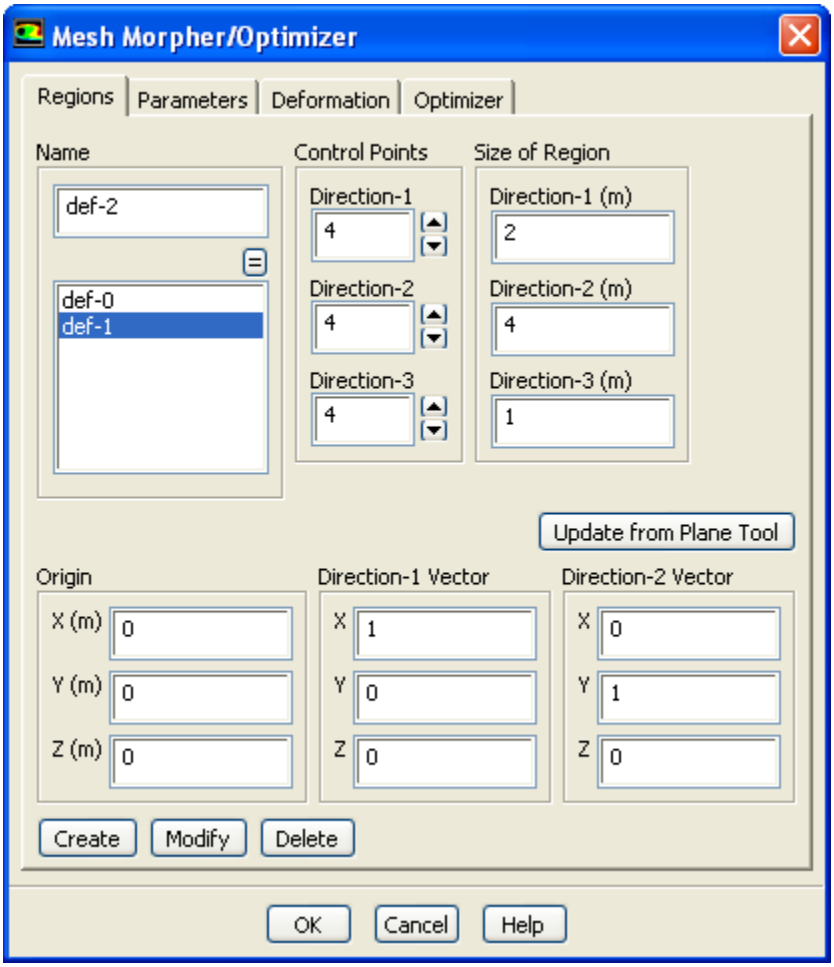
The **Define/Dynamic Mesh...** menu item opens the *Dynamic Mesh Task Page* (p. 1877).

36.3.10. Define/Mesh Morpher/Optimizer...

The **Define/Mesh Morpher/Optimizer...** menu item opens the *Mesh Morpher/Optimizer Dialog Box* (p. 2088).

36.3.10.1. Mesh Morpher/Optimizer Dialog Box

The **Mesh Morpher/Optimizer** dialog box allows you to use the mesh morpher/optimizer to deform the mesh in order to solve shape optimization problems. You can deform the mesh manually, or allow an in-built optimizer to define the deformation in order to minimize an objective function. See *Setting Up the Mesh Morpher/Optimizer* (p. 634) for further details.



Controls

Regions

tab allows you to define the regions of the domain where the mesh will be deformed in order to optimize the shape.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

provides controls for the deformation regions you are defining. The upper text-entry box defines the name of the region that is created when you click the **Create** button. The lower selection list allows you to select a region that has already been created, in order to display it in the graphics window, modify the settings via the **Modify** button, or delete the region via the **Delete** button.

Control Points

allows you to define the number of control points for each axis of the deformation region. The number of control points must be the same for all regions in each direction.

Direction-1

defines the number of control points for the deformation region along the direction specified by the **Direction-1 Vector** group box.

Direction-2

defines the number of control points for the deformation region along the direction specified by the **Direction-2 Vector** group box.

Direction-3

defines the number of control points for the deformation region along the third axis (i.e., the cross product of the vectors defined by the **Direction-1 Vector** and **Direction-2 Vector** group boxes). This number-entry box is only available for 3D cases.

Size of Region

allows you to define the overall dimensions of the deformation region.

Direction-1

defines the length of the deformation region along the direction specified by the **Direction-1 Vector** group box.

Direction-2

defines the length of the deformation region along the direction specified by the **Direction-2 Vector** group box.

Direction-3

defines the length of the deformation region along the third axis (i.e., the cross product of the vectors defined by the **Direction-1 Vector** and **Direction-2 Vector** group boxes). This number-entry box is only available for 3D cases.

Origin

allows you to define the Cartesian coordinates of the origin of the deformation region.

X

defines the *x* coordinate of the origin of the deformation region.

Y

defines the *y* coordinate of the origin of the deformation region.

Z

defines the *z* coordinate of the origin of the deformation region. This number-entry box is only available for 3D cases.

Update from Plane Tool

updates the values in the **Direction-1 Vector** and **Direction-2 Vector** group boxes based on the settings in the plane tool. See *Using the Plane Tool* (p. 1367) for information about the plane tool. This button is only available for 3D cases.

Update from Line Tool

updates the values in the **Direction-1 Vector** and **Direction-2 Vector** group boxes based on the settings in the line tool. See *Using the Line Tool* (p. 1363) for information about the line tool. This button is only available for 2D cases.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction-1 Vector

allows you to define a vector that acts as the first direction of the deformation region relative to the **Origin**.

X

defines the *x* component of the first direction vector of the deformation region.

Y

defines the *y* component of the first direction vector of the deformation region.

Z

defines the *z* component of the first direction vector of the deformation region. This number-entry box is only available for 3D cases.

Direction-2 Vector

defines a vector that is used to calculate the second direction of the deformation region relative to the **Origin**. For 2D cases, the values in the **Direction-2 Vector** group box are automatically defined by ANSYS FLUENT to be perpendicular to the **Direction-1 Vector**. For 3D cases, the vector defined by the **Direction-2 Vector** group box is projected onto a plane that intersects the **Origin** and is perpendicular to the vector defined in the **Direction-1 Vector** group box, in order to calculate the second direction.

X

defines the *x* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only editable for 3D cases.

Y

defines the *y* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only editable for 3D cases.

Z

defines the *z* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only available for 3D cases.

Create

creates a new deformation region with the name specified in the upper text-entry box of the **Name** group box and with the settings defined in the **Control Points**, **Size of Region**, **Origin**, **Direction-1 Vector**, and **Direction-2 Vector** group boxes. The newly created deformation region will be added to and selected in the lower **Name** selection list, and will be displayed in the graphics window.

Modify

modifies the saved settings for the deformation region selected in the lower **Name** selection list and displays the modified region in the graphics window.

Delete

deletes the deformation region selected in the lower **Name** selection list.

Parameters

tab allows you to set the parameters for deformation.

Number of Parameters

defines the number of parameters available to be assigned to control points.

Parameter Index

specifies the index of the parameter whose value is displayed in the **Value** number-entry box. Note that the first parameter has an index of **0**.

Value

defines a magnitude of deformation for the parameter specified in the **Parameter Index** number-entry box when the **Apply** button is clicked. The product of the **Value** and the settings specified in the **Scaling Factors** group box of the **Deformation** tab defines the displacement of a control point

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

during manual deformation. This number-entry box is only editable when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Apply
saves the **Value** you entered for the parameter specified in the **Parameter Index** number-entry box.

Deform
modifies the mesh and updates the mesh display in the graphics window based on the settings saved in the **Parameters** and **Deformation** tabs. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Check
displays a mesh check report in the console for the mesh displayed in the graphics window. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. The mesh check report provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases, in the same manner as the **Check** button in the **General** task page (see *Checking the Mesh* (p. 172) for details).

Deformation
tab allows you to assign deformation parameters to control points.

Region
selects the deformation region for which you are assigning deformation parameters to control points.

Control Point
defines the control point to which you are assigning deformation parameters.

Probe
allows you to specify the **Control Point** by clicking in the graphics window with the **mouse-probe** button of the mouse.

Parameter Index
specifies the index number of the deformation parameter you are assigning to the **Control Point**. Note that the first parameter has an index of **0**.

Scaling Factors
allows you to define coefficients in the *x*, *y*, and (for 3D cases) *z* directions to scale the magnitude of deformation for a given parameter. If you use an in-built optimizer, these coefficients provide the direction of the displacement of the control point and the optimizer determines the overall magnitude of displacement. Alternatively, if you manually specify the deformation, the values you enter in the **Scaling Factors** group box are multiplied with the **Value** specified in the **Parameters** tab to define the displacement of the control point.

X
defines the scaling coefficient applied to the deformation parameter in the *x* direction.

Y
defines the scaling coefficient applied to the deformation parameter in the *y* direction.

Z
defines the scaling coefficient applied to the deformation parameter in the *z* direction. This number-entry box is only available for 3D cases.

Apply
saves the values entered in the **Scaling Factor** group box for the parameter specified in the **Parameter Index** number-entry box, and assigns this parameter to the control point displayed in the **Control Point** number-entry box.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deform

modifies the mesh and updates the mesh display in the graphics window based on the settings saved in the **Parameters** and **Deformation** tabs. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Check

displays a mesh check report in the console for the mesh displayed in the graphics window. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. The mesh check report provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases, in the same manner as the **Check** button in the **General** task page (see *Checking the Mesh* (p. 172) for details).

Reset

undoes any modifications made to the mesh as a result of the **Deform** button and updates the mesh display in the graphics window.

Optimizer

tab allows you to define the settings for the optimizer.

Optimizer

specifies which optimizer is used. The choices include **none** (for manual deformation), **compass**, **powell**, **rosenbrock**, **simplex**, and **torczon**. For information about how the optimizers function, see *Optimizers* (p. 632).

Objective Function Definition

allows you to specify the format in which you have provided the objective function that will be minimized during the optimization process. This group box is not available if **none** is selected from the **Optimizer** drop-down list.

User-Defined Function

specifies that the objective function that will be minimized during the optimization process is provided via a user-defined function.

Scheme Procedure

specifies that the objective function that will be minimized during the optimization process is provided via a Scheme source file.

Optimizer Settings

allows you to specify the optimizer settings. This group box is not available if **none** is selected from the **Optimizer** drop-down list.

Maximum Number of Designs

defines the maximum number of design stages the optimizer will undergo to reach the specified objective function.

Number of Iterations

defines the maximum number of iterations ANSYS FLUENT will perform for each design change.

Optimizer Convergence Criteria

defines the convergence criteria for the optimizer.

Initialization

allows you to specify how the solution variables should be treated after the mesh is deformed during the optimization process. This group box is not available if **none** is selected from the **Optimizer** drop-down list.

Initialize Data After Morphing

specifies that the solution variables should be initialized to the values defined in the **Solution Initialization** task page after deformation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continue with Current Data
specifies that the solution variables should remain the values obtained in the previous design iteration.

Apply
saves the settings in the **Optimizer** tab.

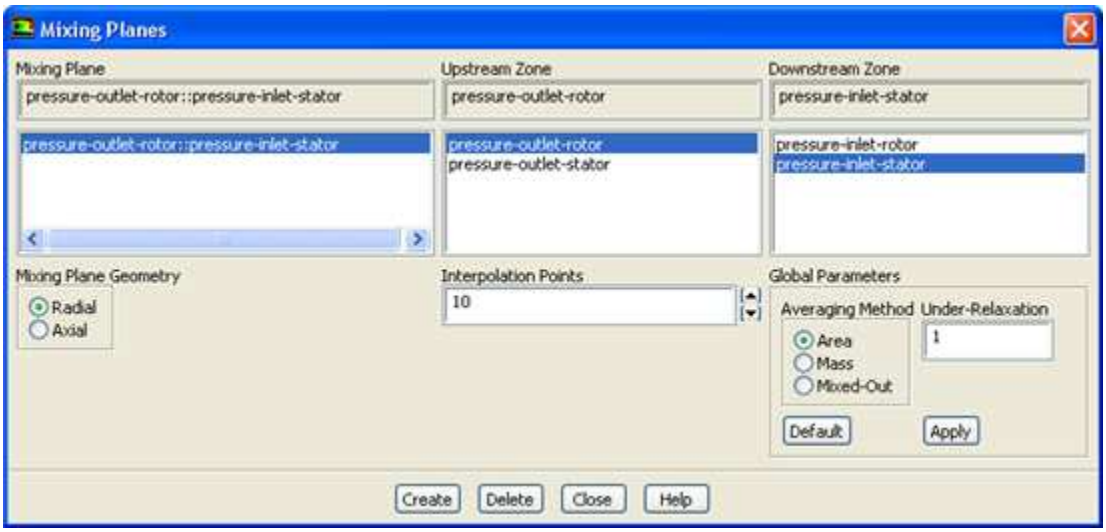
Optimize
initiates the optimization process using the settings saved in all of the tabs of the **Mesh Morpher/Optimizer** dialog box. This button is not available if **none** is selected from the **Optimizer** drop-down list.

36.3.11. Define/Mixing Planes...

The **Define/Mixing Planes...** menu item opens the *Mixing Planes Dialog Box* (p. 2093).

36.3.11.1. Mixing Planes Dialog Box

The **Mixing Planes** dialog box allows you to define the mixing planes for a mixing plane model. See *Setting Up the Mixing Plane Model* (p. 528) for details about using the items below.



Controls

Mixing Plane
contains a list from which you can select an existing mixing plane, and an informational field in which ANSYS FLUENT displays the name of the currently selected (or most recently created) mixing plane.

Upstream Zone, Downstream Zone
contain lists from which you can select the boundaries on the upstream and downstream sides of the mixing plane, and informational fields that show the names of the zone you selected in each list. (You cannot edit these fields; the name in each field will be the name of the zone you selected in the list below it.)

Interpolation Points
specifies the number of radial or axial locations used in constructing the boundary profiles for circumferential averaging. This item appears only in 3D.

Mixing Plane Geometry
defines the geometry of the mixing plane interface. This item appears only in 3D.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radial
specifies that information at the mixing plane interface is to be circumferentially averaged into profiles that vary in the radial direction, e.g., $p(r)$, $T(r)$.

Axial
specifies that circumferentially averaged profiles are to be constructed that vary in the axial direction, e.g., $p(x)$, $T(x)$.

Global Parameters
contains parameters related to the mixing plane calculation.

Averaging Method
consists of three profile averaging methods.

Area
is the default method and is expressed using Equation 2–17 in the Theory Guide.

Mass
provides better representation of the total quantities than the area averaging method. It is defined by Equation 2–18 of the Theory Guide.

Mixed-Out
is most representative of non-uniform flow profiles and is expressed using Equation 2–20 in the Theory Guide.

Under-Relaxation
specifies the under-relaxation factor for updating the boundary values at mixing planes.

Apply
sets the specified **Under-Relaxation**.

Default
sets the **Under-Relaxation** to its default value, as assigned by ANSYS FLUENT. After execution, the **Default** button becomes the **Reset** button.

Reset
resets the **Under-Relaxation** to its most recently saved value (i.e., the value before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

Create
creates the specified mixing plane (and assigns it a name in the **Mixing Plane** field).

Delete
deletes the mixing plane selected under **Mixing Plane**.

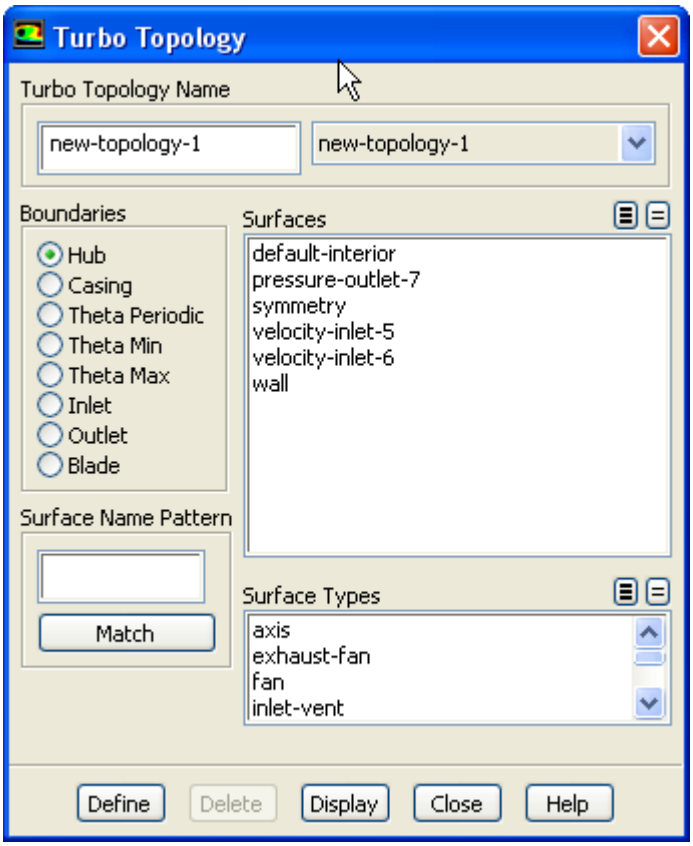
36.3.12. Define/Turbo Topology...

The **Define/Turbo Topology...** menu item opens the *Turbo Topology Dialog Box* (p. 2094).

36.3.12.1. Turbo Topology Dialog Box

The **Turbo Topology** dialog box allows you to define the topology for a turbomachinery application, so that you can use the turbomachinery-specific postprocessing features described in *Turbomachinery Postprocessing* (p. 1483). See *Defining the Turbomachinery Topology* (p. 1484) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Turbo Topology Name

specifies the name of the new topology.

Boundaries

contains radio buttons for the topology boundaries to be defined.

Hub

specifies the definition for the wall zone(s) forming the lower boundary of the flow passage (generally toward the axis of rotation of the machine).

Casing

specifies the definition of the wall zone(s) forming the upper boundary of the flow passage (away from the axis of rotation of the machine).

Theta Periodic

specifies the definition of the periodic boundary zone(s) on the circumferential boundaries of the flow passage.

Theta Min, Theta Max

specify the definition of the wall zones at the minimum and maximum angular (θ) positions on a circumferential boundary.

Inlet

specifies the definition of the inlet zone(s) through which the flow enters the passage.

Outlet

specifies the definition of the outlet zone(s) through which the flow exits the passage.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blade
specifies the definition of the wall zone(s) that defines the blade(s) (if any). Note that these zones cannot be attached to the circumferential boundaries. For this situation, use **Theta Min** and **Theta Max** to define the blade.

Surfaces
contains a selectable list of the available surfaces, from which you can select the surface(s) that represent the boundary selected under **Boundaries**.

Surface Name Pattern
specifies the pattern to look for in the names of surfaces. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Surfaces** list with names that match the specified pattern. See *Defining the Turbomachinery Topology* (p. 1484) for information about matching additional characters using * and ?.

Surface Types
contains a selectable list of types of surfaces. If you select (or deselect) an item in this list, all surfaces of that type will be selected (or deselected) automatically in the **Surfaces** list.

Define
defines the new topology. If you have selected an existing topology the **Define** button is replaced by the **Modify** button.

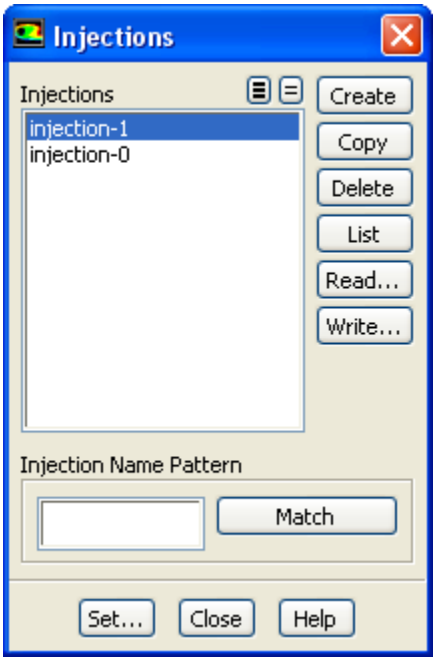
Display
draws the defined topology in the active graphics window.

36.3.13. Define/Injections...

The **Define/Injections...** menu item opens the *Injections Dialog Box* (p. 2096).

36.3.13.1. Injections Dialog Box

The **Injections** dialog box allows you to create, delete, and list discrete phase injections, and access the *Set Injection Properties Dialog Box* (p. 2097) and the *Set Multiple Injection Properties Dialog Box* (p. 2101), in which you can set the properties for the injections. See *Creating and Modifying Injections* (p. 1038) for details.



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Injections
contains a list from which you can select one or more injections in order to set, copy, or modify properties, or delete or list injections.

Create
creates a new injection and opens the *Set Injection Properties Dialog Box* (p. 2097), in which you can set its properties.

Copy
creates a new injection with the same properties as the selected injection and opens the *Set Injection Properties Dialog Box* (p. 2097) where the new injection's properties can be modified.

Delete
deletes the injection(s) selected in the **Injections** list.

List
lists the initial conditions for the particle streams in the injection(s) selected in the **Injections** list.

Read...
opens the *The **Select File** Dialog Box* (p. 33) where you will select the injection file to read in.

Write...
allows you to select the injection from the list and write it to a file.

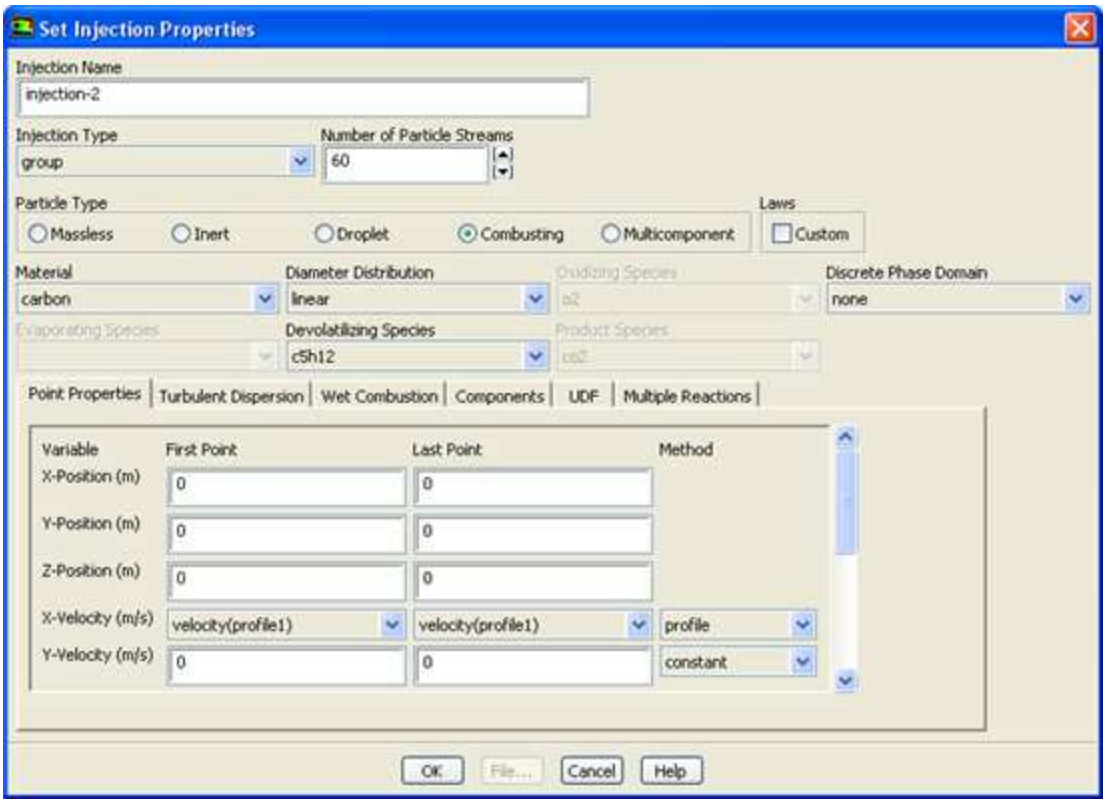
Injection Name Pattern
specifies the pattern to look for in the names of injections. Type the pattern in the text field and click **Match** to select (or deselect) the injections in the **Injections** list with names that match the specified pattern. See *Shortcuts for Selecting Injections* (p. 1040) for information about matching additional characters using * and ?.

Set...
opens the *Set Injection Properties Dialog Box* (p. 2097) for the injection selected in the **Injections** list or the *Set Multiple Injection Properties Dialog Box* (p. 2101) if more than one injection is selected in the **Injections** list.

36.3.13.2. Set Injection Properties Dialog Box

The **Set Injection Properties** dialog box allows you to define the properties of an existing discrete-phase injection (which was created in the *Injections Dialog Box* (p. 2096)). This dialog box is opened from the **Injections** dialog box. See *Defining Injection Properties* (p. 1040) for details about the items listed in this section.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Injection Name

sets the name of the injection.

Injection Type

contains a drop-down list of the available injection types: **single**, **group**, **cone**, **solid-cone**, **surface**, **plain-orifice-atomizer**, **pressure-swirl-atomizer**, **air-blast-atomizer**, **flat-fan-atomizer**, **effervescent-atomizer**, and **file**. (**cone** is not available in 2D.) These choices are described in *Injection Types* (p. 1024).

Number of Particle Streams

indicates the number of particle streams in a **group** or **cone**injection. (This item will not appear for **single**, **surface**, or **file** injections.)

Release From Surfaces

indicates the surface from which the particles in a **surface** injection will be released. (This item will appear only for a **surface** injection.)

Particle Type

specifies the particle type as **Massless**, **Inert**, **Droplet**, **Combusting**, or **Multicomponent**. These types are described in *Particle Types* (p. 1025).

Laws

(not for massless particles) contains inputs for customized particle laws.

Custom

enables the specification of customized particle laws and opens the *Custom Laws Dialog Box* (p. 2102).

Material

(not for massless particles) indicates the material for the particles. If this is the first time you have created a particle of this type, you can choose from all of the materials by copying them from the database or creating them from scratch, as discussed in *Setting Discrete-Phase Physical Properties* (p. 1057) and described in detail in *Using the Materials Task Page* (p. 387).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diameter Distribution

(not for massless particles) allows you to change from the default **linear** interpolation method used to determine the size of the particles in a **group** injection, or the default **uniform** method used to determine the size of the particles in a **surface** injection, to the **rosin-rammler** or **rosin-rammler-logarithmic** method. The Rosin-Rammler method for determining the range of diameters is described in *Using the Rosin-Rammler Diameter Distribution Method* (p. 1035).

Evaporating Species

(for **droplet** particles) specifies the gas-phase species created by the vaporization and boiling laws (laws 2 and 3).

Devolatilizing Species

(for **combusting** particles) specifies the gas-phase species created by the devolatilization law (law 4).

This item will not appear for two-mixture-fraction non-premixed combustion calculations.

Devolatilizing Stream

(for **combusting** particles) specifies the destination stream for the gas-phase species created by the devolatilization law (law 4).

This item will appear only for two-mixture-fraction non-premixed combustion calculations.

Oxidizing Species

(for **combusting** particles) specifies the gas phase species that participates in the surface char combustion reaction (law 5).

Product Species

(for **combusting** particles) specifies the gas-phase species created by the surface char combustion reaction (law 5).

This item will not appear for two-mixture-fraction non-premixed combustion calculations.

Product Stream

(for **combusting** particles) specifies the destination stream for the gas-phase species created by the surface char combustion reaction (law 5).

This item will appear only for two-mixture-fraction non-premixed combustion calculations.

Discrete Phase Domain

is available when using the **Dense Discrete Phase Model**, described in *Including the Dense Discrete Phase Model* (p. 1167).

Point Properties

displays the inputs for the point properties for the injection (e.g., position, velocity, diameter, temperature, and mass flow rate). These inputs are described for each injection type in *Point Properties for Single Injections* (p. 1026) – *Point Properties for Effervescent Atomizer Injections* (p. 1034).

First Point

specifies the first point properties for the injection.

Last Point

specifies the last point properties for the injection.

Turbulent Dispersion

displays the inputs for stochastic tracking and cloud tracking.

Stochastic Tracking

controls the stochastic tracking for turbulent flows. Stochastic tracking includes the effect of turbulent velocity fluctuations on the particle trajectories using the DRW model described in Stochastic Tracking in the Theory Guide. See *Stochastic Tracking* (p. 1044) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Random Walk Model
includes the effect of instantaneous turbulent velocity formulations on the particle trajectories through stochastic method.

Random Eddy Lifetime
specifies that the characteristic lifetime of the eddy is to be random.

Number of Tries
controls the inclusion of turbulent velocity fluctuations.

An input of 1 or greater tells ANSYS FLUENT to include turbulent velocity fluctuations in the particle force balance.

Time Scale Constant
is C_L in Equation 16–16 in the Theory Guide. The default is 0.15; if you use the RSM, a value of 0.3 is recommended.

Cloud Tracking
incorporates the effects of turbulent dispersion on the injection. For details on the following items, see Particle Cloud Tracking in the Theory Guide and *Cloud Tracking* (p. 1045).

Cloud Model
enables particle cloud tracking.

Min. Cloud Diameter
specifies the diameter of the cloud in which the particles enter the domain.

Max. Cloud Diameter
specifies the maximum allowed cloud diameter.

Wet Combustion
displays the inputs for the wet combustion model.

Wet Combustion Model
allows the combusting particles to include an evaporating/boiling material.

Liquid Material
contains a drop-down list of liquid materials that can be chosen as the evaporating/boiling material to be included with the combusting particles.

Liquid Fraction
sets the volume fraction of the liquid present in the particle.

Components
displays the inputs for **Multicomponent** for use in the definition of the particle injection. For details on the following items, see Vapor Liquid Equilibrium Theory in the Theory Guide.

Multicomponent Settings
contains the multicomponent injections.

Component
specifies the component which is a part of the multicomponent species.

Mass Fraction
specifies the mass fraction of the component in a multicomponent species.

Evaporating Species
specifies the gas-phase species to be evaporated.

Evaporating Stream
specifies the source stream from which the species will be evaporated.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF

displays the inputs for **User-Defined Functions** for use in the definition of the particle injection. For details about user-defined functions, see the separate UDF Manual.

Initialization

contains a drop-down list of available user-defined functions. The UDF that you choose will be used to modify the injection properties at the time the particles are injected into the domain.

Heat/Mass Transfer

allows you to select the UDF that defines the heat or mass transfer.

Multiple Reactions

displays the inputs for **Multiple Surface Reactions**. See *Particle Surface Reactions* (p. 835) for details about this model.

Species Mass Fractions

specify the combustible fraction of the combusting particle if you have defined more than one particle surface species. See *Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion* (p. 836) for details.

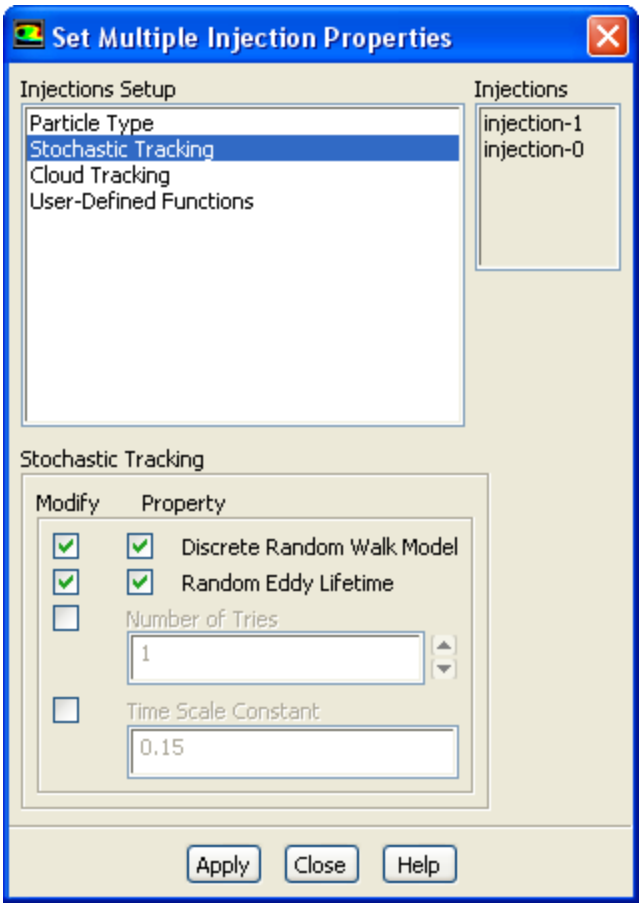
File...

opens the *The **Select File** Dialog Box* (p. 33), in which you can select a file containing the injection definition (when **file** is selected as the **Injection Type**).

36.3.13.3. Set Multiple Injection Properties Dialog Box

The **Set Multiple Injection Properties** dialog box allows you to set properties that are common to multiple injections. This dialog box is opened when you select more than one injections in the *Injections Dialog Box* (p. 2096). See *Defining Properties Common to More than One Injection* (p. 1047) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Injections Setup

contains a list of the categories of injection properties that you can set for the injections in the **Injections** list. These categories correspond to the categories of inputs in the *Set Injection Properties Dialog Box* (p. 2097). When you select an item in the **Injections Setup** list, the dialog box will expand to show the relevant inputs, which are the same as those in the **Set Injection Properties** dialog box.

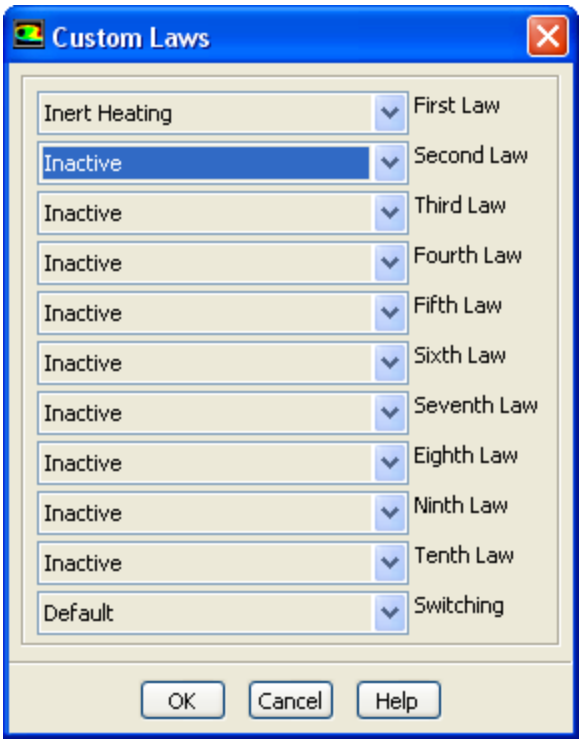
Injections

displays an informational list of the injections for which you are setting common properties. These are the injections that you selected in the *Injections Dialog Box* (p. 2096).

36.3.13.4. Custom Laws Dialog Box

The **Custom Laws** dialog box is used to incorporate user-defined functions (see the separate UDF Manual for details) in place of the default physical laws (1 through 6) used in the heat/mass transfer calculations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

First Law, Second Law, Third Law, Fourth Law, Fifth Law, Sixth Law
contain drop-down lists in which you can choose a user-defined particle law to replace the standard law.

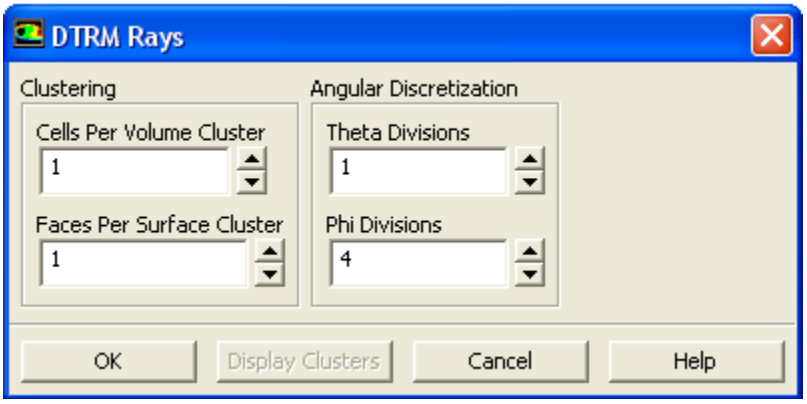
Switching
contains a drop-down list in which you can select a user-defined function that customizes the way ANSYS FLUENT switches between particle laws.

36.3.14. Define/DTRM Rays...

The **Define/DTRM Rays...** menu item opens the *DTRM Rays Dialog Box* (p. 2103).

36.3.14.1. DTRM Rays Dialog Box

The **DTRM Rays** dialog box allows you to define the rays used by the discrete transfer radiation model (DTRM). It opens automatically when you click on **OK** after selecting the **Discrete Transfer** model in the *Radiation Model Dialog Box* (p. 1657). See *Setting Up the DTRM* (p. 707) for details about the items below.



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Clustering
contains parameters for the clusters (see Clustering in the Theory Guide).

Cells Per Volume Cluster, Faces Per Surface Cluster
control the number of radiating surfaces and absorbing cells. (See the explanation in *Controlling the Clusters* (p. 708).)

Angular Discretization
contains parameters for the ray traces (see Ray Tracing in the Theory Guide).

Theta Divisions, Phi Divisions
control the number of rays being traced. (Guidelines are provided in *Controlling the Rays* (p. 708).)

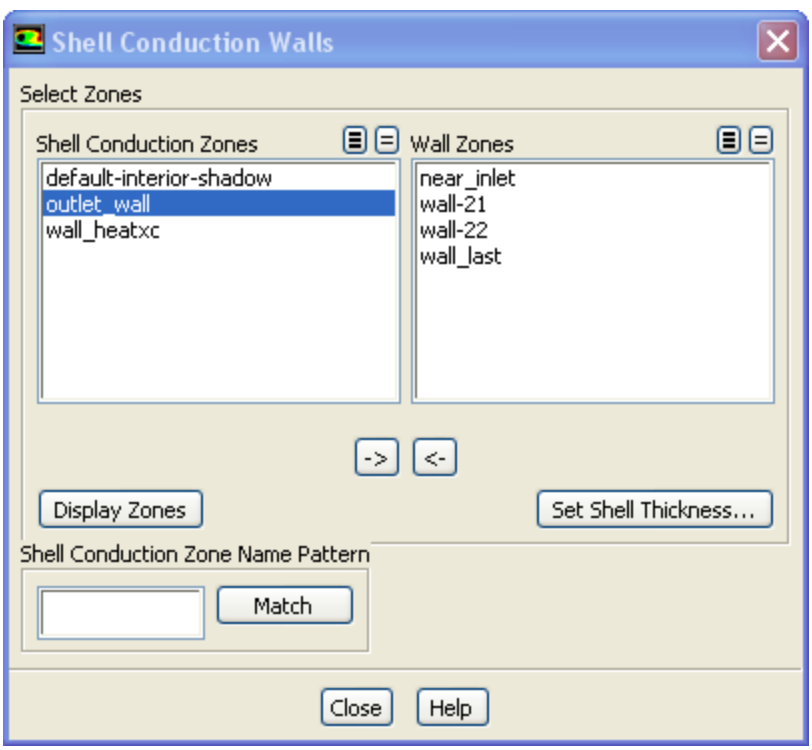
Display Clusters
generates a graphical display of the clusters in the domain. (This item is available only after you have created or read a ray file.)

36.3.15. Define/Shell Conduction Walls...

The **Define/Shell Conduction Walls...** menu item opens the *Shell Conduction Walls Dialog Box* (p. 2104).

36.3.15.1. Shell Conduction Walls Dialog Box

The **Shell Conduction Walls** dialog box allows you to manage, define, and display shell conduction zones all in one location. See *Managing Shell Conduction Walls* (p. 703) for details about using the **Shell Conduction Walls** dialog box.



Controls

Select Zones
contains the list of **Shell Conduction Zones** and **Wall Zones**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In essence, the −−> button disables shell conduction and the <−− button enables shell conduction.

Shell Conduction Zones
contains a list of zones with shell conduction enabled.

Wall Zones
contains a list of zones without shell conduction.

−−>, <−−
disables and enables shell conduction, respectively.

Display Zones
displays the selected wall(s) in the graphics window. Note that you can select walls with or without shell conduction. The zones will be displayed with different colors depending on the option selected in *Mesh Colors Dialog Box* (p. 1638) (accessible from the *Mesh Display Dialog Box* (p. 1635)).

Set Shell Thickness...
opens the *Set Shell Thickness Dialog Box* (p. 2105)

Shell Conduction Zone Name Pattern
specifies the pattern to look for in the names of shell conduction zones. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Shell Conduction Zones** list with names that match the specified pattern.

36.3.15.2. Set Shell Thickness Dialog Box

The **Set Shell Thickness** dialog box allows you to set the thicknesses of the selected **Shell Conduction Zones** in the *Shell Conduction Walls Dialog Box* (p. 2104). See *Managing Shell Conduction Walls* (p. 703) for details about the items below.

Controls

Thickness
allows you to apply the same thickness to all the selected shell conduction zones. Click the **Apply** button to accept the specified thickness.

Shell Conduction Zones
allows you to apply different thicknesses to the selected shell conduction zones; enter the values against each zone listed in the **Shell Conduction Zones** group box.

36.3.16. Define/Custom Field Functions...

The **Define/Custom Field Functions...** menu item opens the *Custom Field Function Calculator Dialog Box* (p. 2106).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

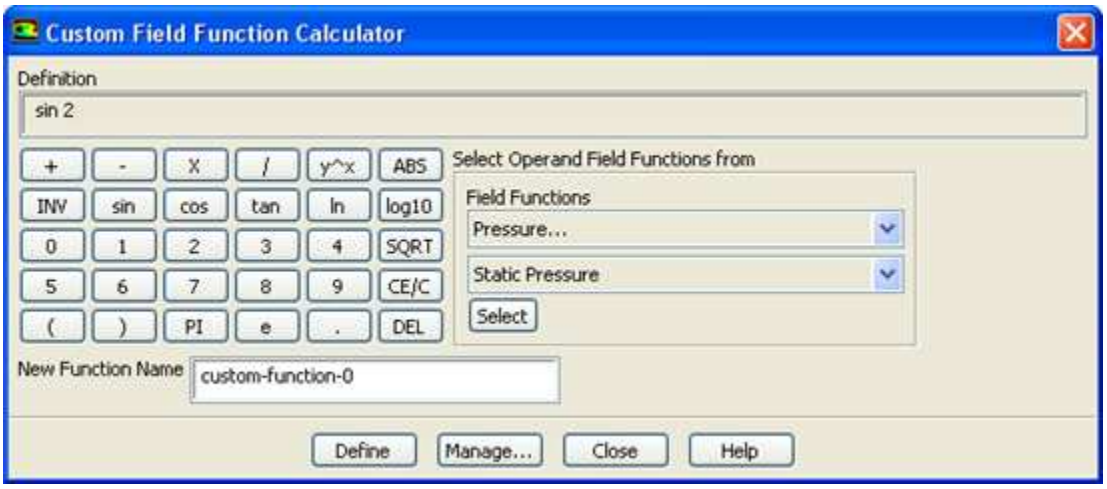
36.3.16.1. Custom Field Function Calculator Dialog Box

The **Custom Field Function Calculator** dialog box allows you to define custom field functions based on existing functions, using simple calculator operators. Any functions that you define will be added to the list of default flow variables and other field functions provided by ANSYS FLUENT.

Important

Recall that you must enter all constants in the function definition in SI units.

See *Creating a Custom Field Function* (p. 1580) for details about the items below.



Controls

Definition
displays the function that you are currently defining. As you select each item from the **Field Functions** list or the calculator keypad, it will appear in the **Definition** text entry box. You *cannot* edit the contents of this box directly; if you want to delete part of a function, use the **Delete** button on the keypad.

(Calculator Buttons)
are push buttons that perform calculator operations. When you select a calculator button (by clicking on it), the appropriate symbol will appear in the **Definition** text entry box.

Select Operand Field Functions from
contains the available field functions and the means for selecting them.

Field Functions
contains a list from which you can select a variable to be used in the definition of a new function.

Select
enters the variable that is currently selected in the **Field Functions** list in the **Definition** field.

New Function Name
specifies the name of the function you are defining. Should you decide to change the function name after you have defined the function, you can do so in the *Field Function Definitions Dialog Box* (p. 2107), which you can open by clicking on the **Manage...** push button.

Define
creates the function and adds it to the list of **Custom Field Functions** within the drop-down list of available field functions. The **Define** push button is grayed out after you create a new function or if the **Definition** field is empty.

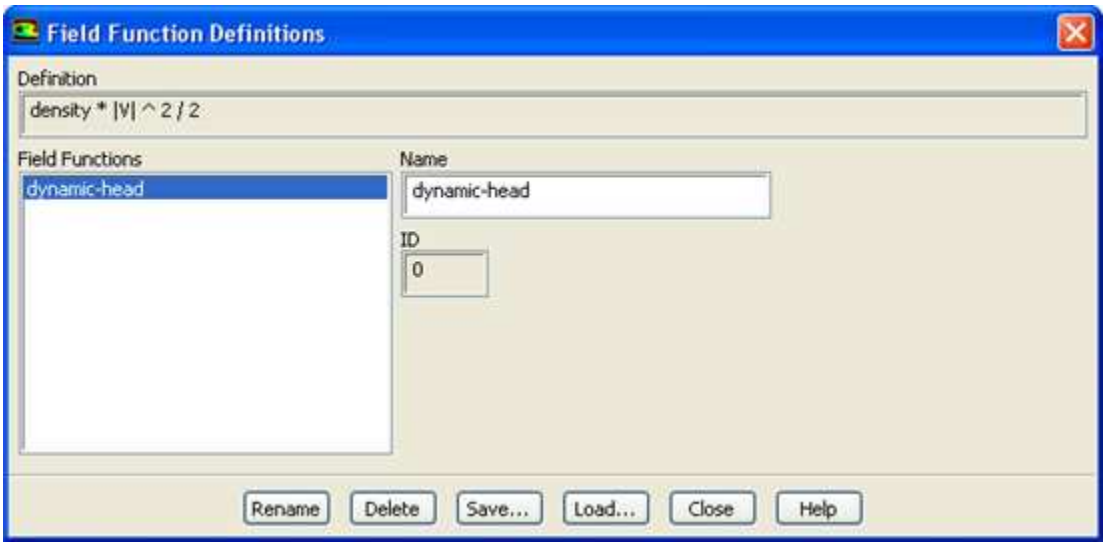
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Manage...

opens the *Field Function Definitions Dialog Box* (p. 2107), which enables you to check, rename, save, load, and delete custom field functions.

36.3.16.2. Field Function Definitions Dialog Box

The **Field Function Definitions** dialog box allows you to check, rename, save, load, and delete custom field functions that you defined in the *Custom Field Function Calculator Dialog Box* (p. 2106). See *Manipulating, Saving, and Loading Custom Field Functions* (p. 1582) for details about the items below.



Controls

Definition

displays the function selected in the **Field Functions** list. This display is for informational purposes only; you cannot edit it.

Field Functions

contains a selectable list of custom field functions. When you select a function, its definition will appear in the **Definition** box and its name will appear in the **Name** text entry box.

Name

displays the name of the currently selected field function. You can enter a new name in this box if you want to rename the function.

ID

reports the ID number of the selected function. The field function at the top of the list has an ID of 0, the second function has an ID of 1, and so on.

Rename

changes the name of the selected function to the name specified in the **Name** text entry box.

Delete

deletes the selected field function.

Save...

opens the *The **Select File** Dialog Box* (p. 33), where you can specify a file in which to save all of the functions in the **Field Functions** list.

Load...

opens the **Select File** dialog box, where you can specify a file from which to read custom field functions (i.e., a file that you saved using the **Save...** button above).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.17. Define/Parameters...

The **Define/Parameters...** menu item opens the *Parameters Dialog Box* (p. 2041).

36.3.18. Define/Profiles...

The **Define/Profiles...** menu item opens the *Profiles Dialog Box* (p. 1808).

36.3.19. Define/Units...

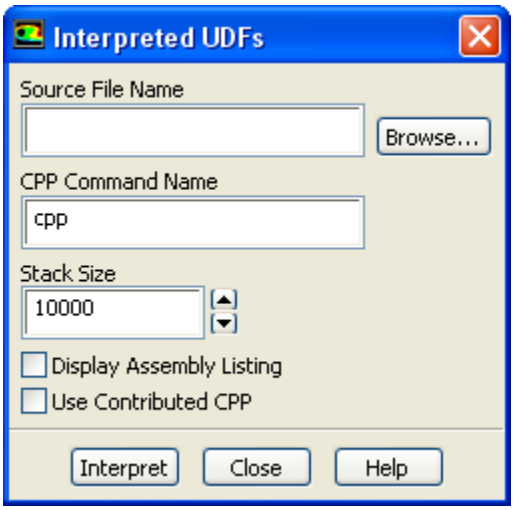
The **Define/Units...** menu item opens the *Set Units Dialog Box* (p. 1637).

36.3.20. Define/User-Defined/Functions/Interpreted...

The **Define/User-Defined/Functions/Interpreted...** menu item opens the *Interpreted UDFs Dialog Box* (p. 2108).

36.3.20.1. Interpreted UDFs Dialog Box

The **Interpreted UDFs** dialog box allows you to compile user-defined functions. See the separate UDF Manual for details.



Controls

Source File Name
sets the name of your user-defined function.

CPP Command Name
sets the name of your C preprocessor.

Stack Size
sets the size of the stack. Keep the default **Stack Size** setting of 10000, unless the number of local variables in your function will cause the stack to overflow. In this case, set the **Stack Size** to a number that is greater than the number of local variables used.

Display Assembly Listing
indicates whether or not to display a listing of assembly language code in your console window as the function compiles.

Use Contributed CPP
specifies the use of the C preprocessor that Fluent Inc. has supplied, instead of your own.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

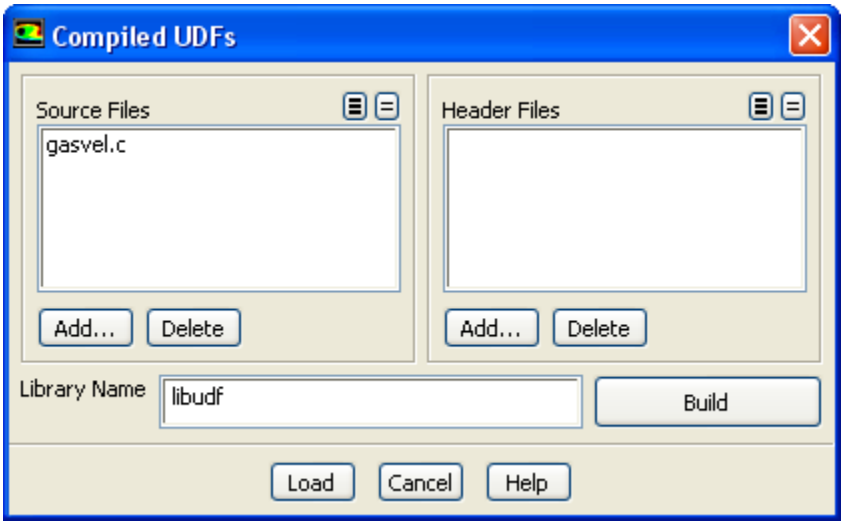
Interpret
interprets the specified function.

36.3.21. Define/User-Defined/Functions/Compiled...

The **Define/User-Defined/Functions/Compiled...** menu item opens the *Compiled UDFs Dialog Box* (p. 2109).

36.3.21.1. Compiled UDFs Dialog Box

The **Compiled UDFs** dialog box allows you to open a library of compiled user-defined functions. See the separate UDF Manual for details.



Controls

Source Files
contains a list of source files.

Header Files
contains a list of header files.

Add...
opens the **Select File** dialog box.

Delete
deletes the selected file from the list.

Library Name
specifies the name of the library to be created.

Build
builds the library and compiles the UDF.

Load
opens the specified library and loads the UDF.

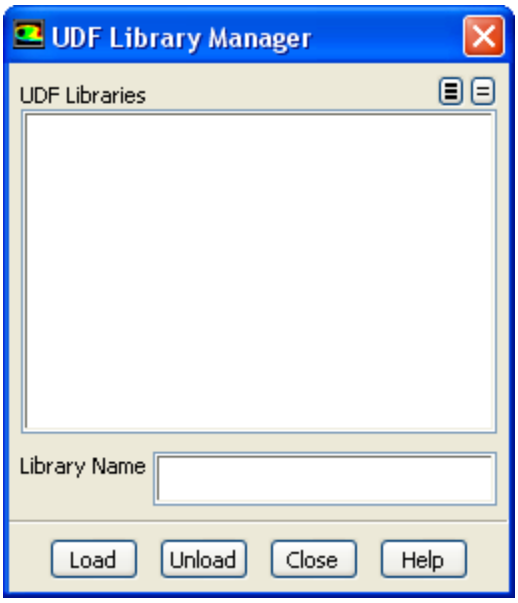
36.3.22. Define/User-Defined/Functions/Manage...

The **Define/User-Defined/Functions/Manage...** menu item opens the *UDF Library Manager Dialog Box* (p. 2110).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.22.1. UDF Library Manager Dialog Box

The **UDF Library Manager** dialog box allows you to load/unload the UDF libraries. See the separate UDF Manual for details.



Controls

UDF Libraries
lists the UDF libraries that are loaded in ANSYS FLUENT.

Library Name
specifies the name of the library to be loaded/unloaded.

Load
opens the specified library and loads the UDF.

Unload
unloads the specified library.

36.3.23. Define/User-Defined/Function Hooks...

The **Define/User-Defined/Function Hooks...** menu item opens the *User-Defined Function Hooks Dialog Box* (p. 2110).

36.3.23.1. User-Defined Function Hooks Dialog Box

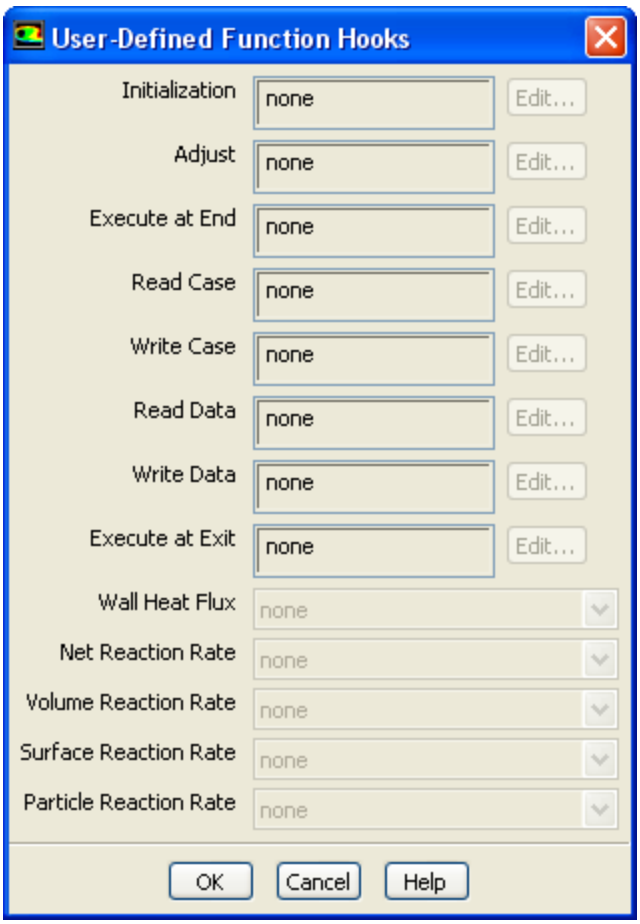
The **User-Defined Function Hooks** dialog box allows you to specify user-defined functions (UDFs) connected to a number of models and procedures in ANSYS FLUENT. See the separate UDF Manual for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

You can hook multiple UDFs to the following functions:

- **Controls**
- **Initialization**
- **Adjust**
- **Execute At End**
- **Read Case**
- **Write Case**
- **Read Data**
- **Write Data**
- **Execute at Exit**



Controls

Initialization

selects a UDF that is called immediately after you initialize your flow field.

Adjust

selects a UDF that is called at the beginning of an iteration before solution of velocities, pressure, and other quantities begins.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Execute At End
selects a UDF that is called at the end of an iteration or time step.

Read Case
selects a UDF that defines a customized section that is to be read from the case file.

Write Case
selects a UDF that defines a customized section that is to be written to the case file.

Read Data
selects a UDF that defines a customized section that is to be read from the data file.

Write Data
selects a UDF that defines a customized section that is to be written to the data file.

Execute at Exit
selects a UDF that is called when exiting an ANSYS FLUENT session.

Wall Heat Flux
selects a UDF that modifies the way that the solver computes the heat flux between a wall and the neighboring fluid cells.

Net Reaction Rate
selects a UDF that defines the net reaction rate.

Volume Reaction Rate
selects a UDF that defines a volumetric reaction rate.

Surface Reaction Rate
selects a UDF that defines a surface reaction rate.

Particle Reaction Rate
selects a UDF that defines a particle reaction rate.

Turbulent Premixed Source
selects a UDF that defines the turbulent flame speed and source term for the premixed or partially premixed combustion model.

Chemistry Step
selects a UDF that defines the chemistry step function.

Spray Collide
selects a UDF that defines the spray collide function.

Cavitation Mass Rate
selects a UDF that defines the cavitation rate.

DO Source
selects a UDF that defines the discrete ordinate source function.

DO Diffuse Reflectivity
selects a UDF that defines the diffuse reflectivity function for the DO radiation model.

DO Specular Reflectivity
selects a UDF that defines the specular reflectivity function for the DO radiation model.

Emissivity Weighting Factor

selects a UDF that defines the emissivity weighting factor for the non-gray DO radiation model or the non-gray P-1 radiation model.

Thickened Flame Model Parameters

selects a UDF that defines the parameters for the Thickened Flame Model.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.24. Define/User-Defined/Execute on Demand...

The **Define/User-Defined/Execute on Demand...** menu item opens the *Execute on Demand Dialog Box* (p. 2113).

36.3.24.1. Execute on Demand Dialog Box

The **Execute on Demand** dialog box allows you to execute a specified user-defined function immediately. See the separate UDF Manual for details.

Controls

Execute on Demand
selects the UDF to be executed.

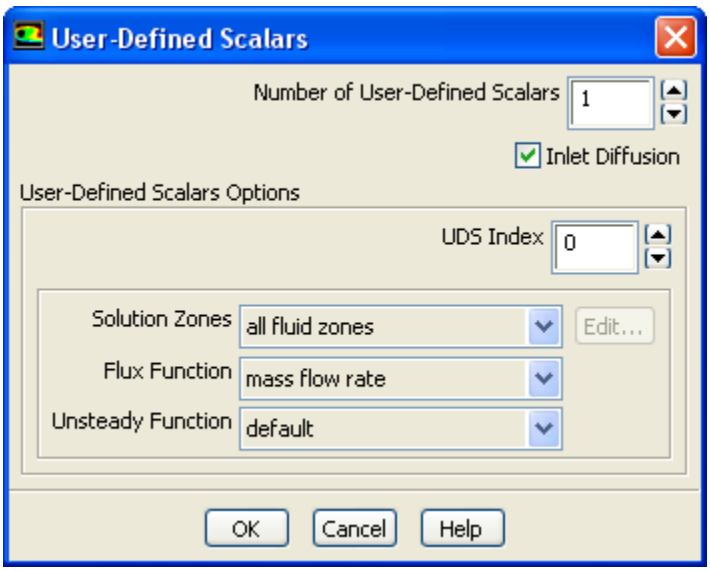
Execute
executes the selected function.

36.3.25. Define/User-Defined/Scalars...

The **Define/User-Defined/Scalars...** menu item opens the *User-Defined Scalars Dialog Box* (p. 2113).

36.3.25.1. User-Defined Scalars Dialog Box

The **User-Defined Scalars** dialog box allows you to include user-defined scalar transport equations in your calculation. See *User-Defined Scalar (UDS) Transport Equations* (p. 485) for details.



Controls

Number of User-Defined Scalars
specifies how many additional scalar transport equations you would like to include in the calculation.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inlet Diffusion
when enabled allows you to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

User-Defined Scalars Options
contains settings that define the scalar transport equation.

UDS Index
when set to 0 marks the first user-defined scalar equation.

Solution Zones
specifies in which zone the scalar equation will be solved: **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**.

Flux Function
is a drop-down list containing available functions for the convection term of the user-defined scalar transport equation(s).

none
(the default) indicates that there is no convection term included in the scalar transport equation(s); i.e., you want to solve a Poisson equation instead of a convection/diffusion equation.

mass flow rate
indicates that the convection term in the scalar transport equation(s) is equal to the mass flow rate $\rho \vec{v} \cdot \vec{A}$.

Note that the **none** and **mass flow rate** options will apply to all solved user-defined scalars. A user-defined flux function must be supplied if a different convective flux is desired for each user-defined scalar.

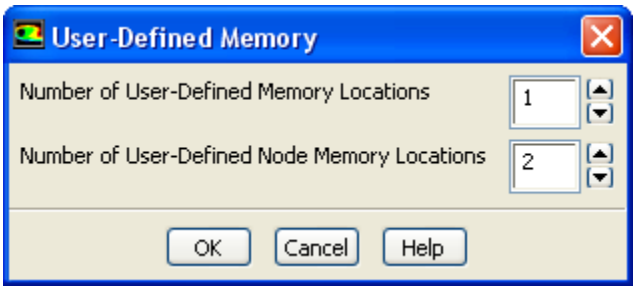
Unsteady Function
is a drop-down list containing available functions for the unsteady term of the user-defined scalar transport equation(s).

36.3.26. Define/User-Defined/Memory...

The **Define/User-Defined/Memory...** menu item opens the *User-Defined Memory Dialog Box* (p. 2114).

36.3.26.1. User-Defined Memory Dialog Box

The **User-Defined Memory** dialog box allows you to allocate memory for user-defined storage variables. See the separate UDF Manual for details.



Controls

Number of User-Defined Memory Locations
specifies the number of memory locations to be allocated.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of User-Defined Node Memory Locations
specifies the number of node memory locations to be allocated.

Important

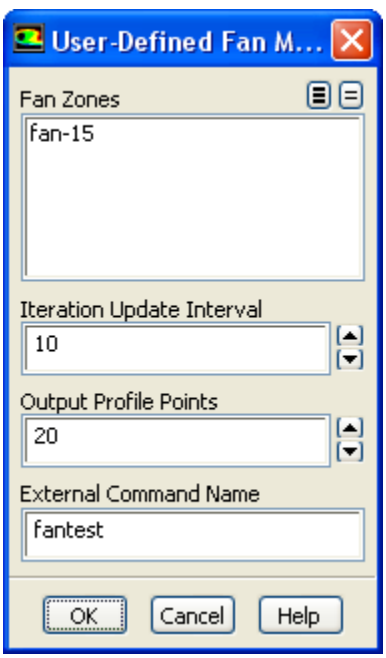
For postprocessing **User-Defined Memory** in CFD-Post, the ANSYS FLUENT user must select the UDM quantities using the **Data File Quantities** option and subsequently write the data file to postprocess the quantities in ANSYS FLUENT. Alternatively, you have the option of exporting the desired quantities to a.cdat file. This will ensure that all the UDM quantities are available for postprocessing in CFD-Post. Please refer to the ANSYS CFD-Post manual for more information.

36.3.27. Define/User-Defined/Fan Model...

The **Define/User-Defined/Fan Model...** menu item opens the *User-Defined Fan Model Dialog Box* (p. 2115).

36.3.27.1. User-Defined Fan Model Dialog Box

The **User-Defined Fan Model** dialog box allows you to periodically regenerate a profile file which can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan. See *User-Defined Fan Model* (p. 359) for details about this feature and how to use this dialog box.



Controls

Fan Zones
contains a list from which you can select the fan zone(s) on which your executable will operate.

Iteration Update Interval
specifies how often the executable will be called on to update the fan profile file.

Output Profile Points
specifies the number of points in the profile file to be written by ANSYS FLUENT.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

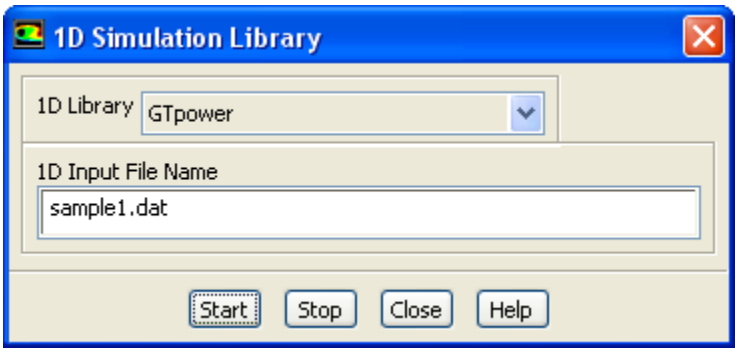
External Command Name
specifies the name of the executable.

36.3.28. Define/User-Defined/1D Coupling...

The **Define/User-Defined/1D Coupling...** menu item opens the *1D Simulation Library Dialog Box* (p. 2116).

36.3.28.1. 1D Simulation Library Dialog Box

The **1D Simulation Library** dialog box allows you to set parameters related to coupling between ANSYS FLUENT and GT-Power or WAVE. See *Coupling Boundary Conditions with GT-Power* (p. 379) or *Coupling Boundary Conditions with WAVE* (p. 381) for details about the items below.



Controls

1D Library
specifies the type of library to be used. (Currently only **GTpower** and **WAVE** are available.)

1D Input File Name
specifies the name of the GT-Power or WAVE input file.

Start
starts up GT-Power or WAVE and generates ANSYS FLUENT user-defined functions for each boundary in the input file.

Stop
unlinks the shared library.

36.4. Solve Menu

For additional information, please see the following sections:

36.4.1. Solve/Methods...
36.4.2. Solve/Controls...
36.4.3. Solve/Monitors...
36.4.4. Solve/Initialization...
36.4.5. Solve/Calculation Activities...
36.4.6. Solve/Run Calculation....

36.4.1. Solve/Methods...

The **Solve/Methods...** menu item opens the *Solution Methods Task Page* (p. 1900).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.4.2. Solve/Controls...

The **Solve/Controls...** menu item opens the *Solution Controls Task Page* (p. 1902).

36.4.3. Solve/Monitors...

The **Solve/Monitors...** menu item opens the *Monitors Task Page* (p. 1913).

36.4.4. Solve/Initialization...

The **Solve/Initialization...** menu item opens the *Solution Initialization Task Page* (p. 1937).

36.4.5. Solve/Calculation Activities...

The **Solve/Calculation Activities...** menu item opens the *Calculation Activities Task Page* (p. 1942).

36.4.6. Solve/Run Calculation....

The **Solve/Run Calculation...** menu item opens the *Run Calculation Task Page* (p. 1955).

36.5. Adapt Menu

For additional information, please see the following sections:

36.5.1. Adapt/Boundary...
36.5.2. Adapt/Gradient...
36.5.3. Adapt/Iso-Value...
36.5.4. Adapt/Region...
36.5.5. Adapt/Volume...
36.5.6. Adapt/Yplus/Ystar...
36.5.7. Adapt/Anisotropic...
36.5.8. Adapt/Manage...
36.5.9. Adapt/Controls...
36.5.10. Adapt/Geometry...
36.5.11. Adapt/Display Options...
36.5.12. Adapt/Smooth/Swap...

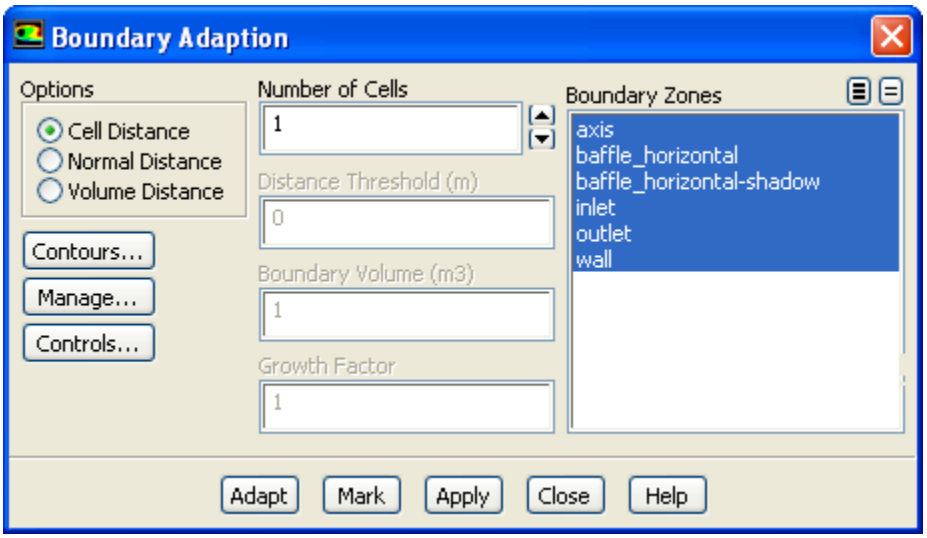
36.5.1. Adapt/Boundary...

The **Adapt/Boundary...** menu item opens the *Boundary Adaption Dialog Box* (p. 2117)

36.5.1.1. Boundary Adaption Dialog Box

The **Boundary Adaption** dialog box allows you to mark or refine boundary cells on selected boundary zones. See *Boundary Adaption* (p. 1328) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains three different methods for boundary adaption:

Cell Distance

enables adaption based on a cell's distance from the boundary, measured in number of cells. See *Boundary Adaption Based on Number of Cells* (p. 1329) for details.

Normal Distance

enables adaption based on a cell's normal distance from the boundary. See *Boundary Adaption Based on Normal Distance* (p. 1330) for details.

Volume Distance

enables adaption based on a target boundary volume and growth factor. See *Boundary Adaption Based on Target Boundary Volume* (p. 1330) for details.

Contours...

opens the *Contours Dialog Box* (p. 1967), which you can use to determine the appropriate parameters for the boundary adaption.

Manage...

opens the *Manage Adaption Registers Dialog Box* (p. 2129), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...

opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Number of Cells

sets the maximum boundary cell distance for adaption (used with the **Cell Distance** option.

Distance Threshold

sets the maximum normal distance for adaption (used with the **Normal Distance** option).

Boundary Volume

sets the boundary volume $V_{boundary}$ in *Equation 29–1* (p. 1331) (used with the **Volume Distance** option).

Growth Factor

sets the exponential growth factor α in *Equation 29–1* (p. 1331) (used with the **Volume Distance** option).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Zones

contains a selectable list of zones on which you can refine. The boundary cells associated with the zones that you select will be refined.

Adapt

refines the cells with edges/faces on the zones selected in the **Boundary Zones** list.

Mark

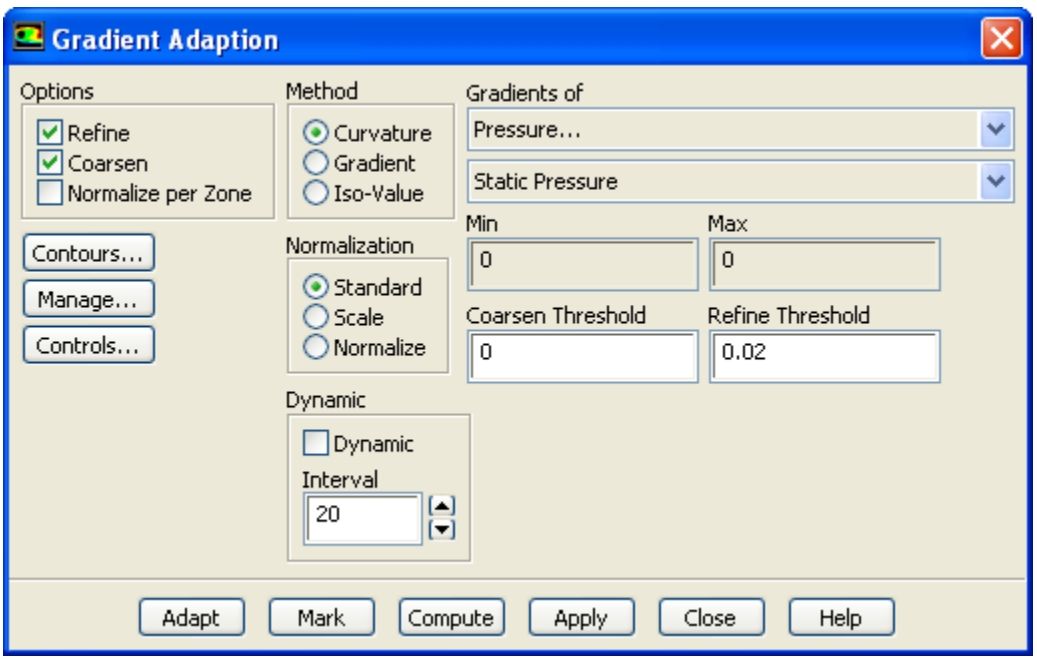
marks the boundary cells associated with the zones selected in the **Boundary Zones** list for refinement. This command produces an adaption register. For information on using adaption registers, see *Manipulating Adaption Registers* (p. 1343).

36.5.2. Adapt/Gradient...

The **Adapt/Gradient...** menu item opens the *Gradient Adaption Dialog Box* (p. 2119).

36.5.2.1. Gradient Adaption Dialog Box

The **Gradient Adaption** dialog box allows you to mark or adapt to gradients of a specified field variable. See *Gradient Adaption* (p. 1331) for details.



Controls

Options

contains the check buttons that toggle the ability to mark and/or adapt cells for refinement or coarsening.

Refine

toggles the ability to refine or mark cells for refinement.

Coarsen

toggles the ability to coarsen or mark cells for coarsening (available in 2D and axisymmetric cases).

Normalize per Zone

toggles the ability of zonal normalization.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contours...
opens the *Contours Dialog Box* (p. 1967), which you can use as an aid to selecting adaption thresholds by displaying contours of adaption function.

Manage...
opens the *Manage Adaption Registers Dialog Box* (p. 2129), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Method
contains options for specifying the criterion for adaption.

Curvature
specifies the use of the second gradient of a field variable for adaption. This approach is recommended for problems with smooth solutions.

Gradient
specifies the use of the first gradient of a field variable for adaption. This approach is recommended for problems with strong shocks.

Iso-Value
allows you to customize the adaption criterion (using custom field functions, user-defined scalars, etc.).

Normalization
contains the options available for normalization.

Standard
specifies that the gradient or curvature is not normalized.

Scale
specifies that the gradient or curvature is scaled by its average value in the domain.

Normalize
specifies that the gradient or curvature is scaled by its maximum value in the domain (i.e. the gradient or curvature is bounded by [0, 1]).

Dynamic
contains options to specify dynamic gradient adaption.

Dynamic
enables dynamic gradient adaption.

Interval
allows you to specify the number of iterations or time-steps between two consecutive automatic mesh adaptions, depending on whether you are performing a steady-state or a time-dependent solution, and on which solver you are using.

Gradients of
contains a list of the field variables that can be used in the gradient adaption function.

Min/Max
displays the minimum and maximum cell values of the gradient adaption function based on the selected quantity.

Coarsen Threshold
designates the threshold values for coarsening the mesh. Cells with adaption function values below the **Coarsen Threshold** will be marked for coarsening.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refine Threshold
designates the threshold values for refining the mesh. Cells with adaption function values above the **Refine Threshold** will be marked for refinement.

Adapt
adapts the mesh based on the gradients of the selected scalar quantity, the coarsening and refining toggle buttons and thresholds, and the adaption limits.

Mark
marks cells to be refined and/or coarsened based on the gradients of the selected quantity and the coarsening and refining toggle buttons and thresholds. This command produces an adaption register. For information on using adaption registers, see *Manipulating Adaption Registers* (p. 1343).

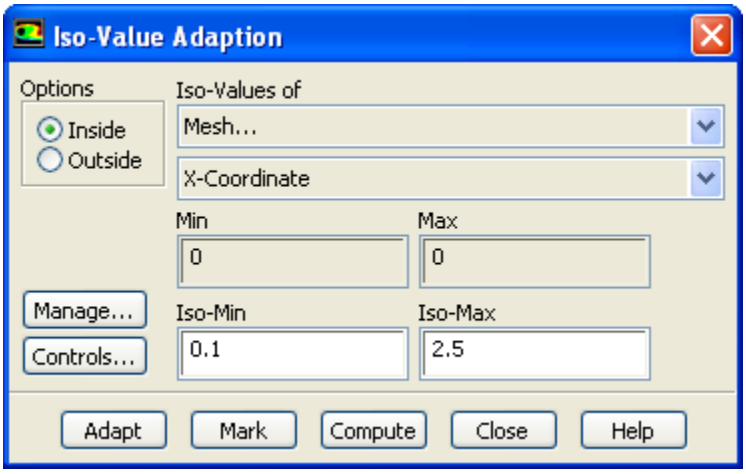
Compute
computes the minimum and maximum cell values of the gradient adaption function with the selected scalar quantity. The values are displayed in the **Min** and **Max** real number fields.

36.5.3. Adapt/Iso-Value...

The **Adapt/Iso-Value...** menu item opens the *Iso-Value Adaption Dialog Box* (p. 2121).

36.5.3.1. Iso-Value Adaption Dialog Box

The **Iso-Value Adaption** dialog box allows you to mark or refine cells inside or outside a specified range of a selected scalar function. See *Isovalue Adaption* (p. 1335) for details.



Controls

Options
contains radio buttons that control whether the cells inside or outside the isovalue range are marked for refinement.

Inside
enables the marking of cells with values between **Iso-Min** and **Iso-Max**.

Outside
enables the marking of cells with values less than **Iso-Min** or greater than **Iso-Max**.

Iso-Values of
contains a list from which you can select the solution variable to be used in the isovalue adaption function.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min/Max
displays the minimum and maximum cell values of the selected field variable. The real number field values are not editable; they are purely informational.

Iso-Min
defines the minimum isovalue threshold.

Iso-Max
defines the maximum isovalue threshold.

Manage...
opens the *Manage Adaption Registers Dialog Box* (p. 2129), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Adapt
adapts the mesh based on the isovalues of the selected solution variable, the isovalue ranges, and the **Inside/Outside** option.

Mark
marks cells to be refined based on the isovalues of the selected quantity, the isovalue ranges, and the **Inside/Outside** option. This command produces an adaption register. For information on using adaption registers, see *Manipulating Adaption Registers* (p. 1343).

Compute
computes the minimum and maximum cell values of the selected solution variable and displays them in the **Min** and **Max** real number fields.

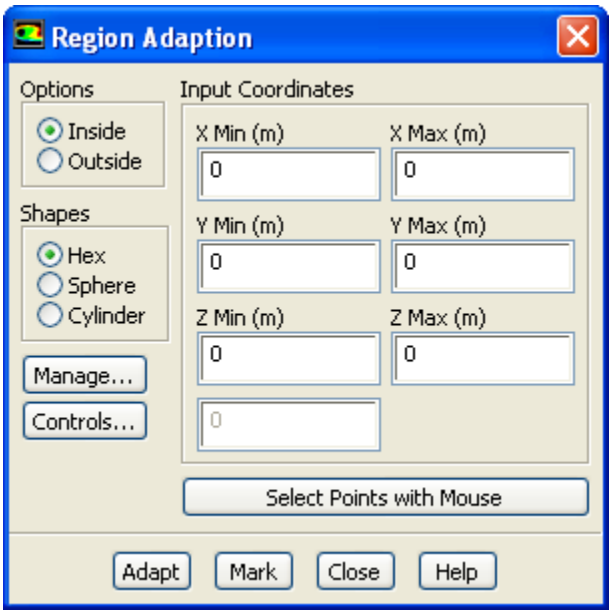
36.5.4. Adapt/Region...

The **Adapt/Region...** menu item opens the *Region Adaption Dialog Box* (p. 2122).

36.5.4.1. Region Adaption Dialog Box

The **Region Adaption** dialog box allows you to mark or refine cells inside or outside a specified region defined by text or mouse input. See *Region Adaption* (p. 1336) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains radio buttons that control whether the cells inside or outside the region are marked for refinement.

Inside

enables the marking of cells with centroids that are within the region.

Outside

enables the marking of cells with centroids that are outside the region.

Shapes

contains radio buttons that control the type of region.

Hex/Quad

defines a hexahedral region in 3D or a rectangular region in 2D. The appropriate button will appear for the solver you are using.

Sphere/Circle

defines a spherical region in 3D or a circular region in 2D. The appropriate button will appear for the solver you are using.

Cylinder

defines a cylindrical region in 3D or a rectangular region in 2D.

Manage...

opens the *Manage Adaption Registers Dialog Box* (p. 2129), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...

opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Input Coordinates

defines the extent of the selected region. The appearance of this box changes depending on the type of region selected.

If the region selected is a hexahedron or quadrilateral, you will input the minimum and maximum coordinates defining the box. (The **Radius** real number field will not be active.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Min, Y Min, Z Min
define the coordinates of the minimum point defining the hexahedron or rectangle. For quadrilaterals, the **Z Min** real entry field will not be active.

X Max, Y Max, Z Max
define the coordinates of the maximum point defining the hexahedron or rectangle. For quadrilaterals, the **Z Max** real entry field will not be active.

If the region selected is a sphere or circle, you will input the coordinates of the sphere's center and its radius. (The maximum coordinate real number fields will not be active.)

X Center, Y Center, Z Center
are the coordinates of the centroid of the sphere or circle. For circles, the **Z Center** real entry field will not be active.

Radius
is the radius of the sphere or circle.

If the region selected is a cylinder, you will input the minimum and maximum coordinates defining the cylinder axis, as well as the radius of the cylinder.

X-Axis Min, Y-Axis Min, Z-Axis Min
define the coordinates of the minimum point defining the cylinder axis. For 2D cases, the **Z-Axis Min** real entry field will not be active.

X-Axis Max, Y-Axis Max, Z-Axis Max
define the coordinates of the maximum point defining the cylinder axis. For 2D cases, the **Z-Axis Max** real entry field will not be active.

Radius
is the radius of the cylinder. (In 2D, this will be the width of the resulting rectangle.)

Select Points with Mouse
activates selection of input coordinates with the mouse. If one of the mouse buttons is defined as a mouse probe, you may select the input coordinates from a display of the mesh or solution field. For more information on mouse buttons, please refer to *Controlling the Mouse Button Functions* (p. 1427). After you select the points, the values will be loaded automatically into the appropriate real number field. If you desire, you can edit these values before marking or adapting. The order of input for defining a hexahedron (rectangle) is insignificant, but the order of input for the sphere (circle) has significance. First, you select the location of the centroid. Then you select a point that lies on the sphere, i.e., a point that is one radius away from the centroid.

Adapt
adapts the mesh based on the region defined and the in/out option.

Mark
marks the cells to be refined based on the region defined and the in/out option. This command produces an adaption register. For information on using adaption registers, see *Manipulating Adaption Registers* (p. 1343).

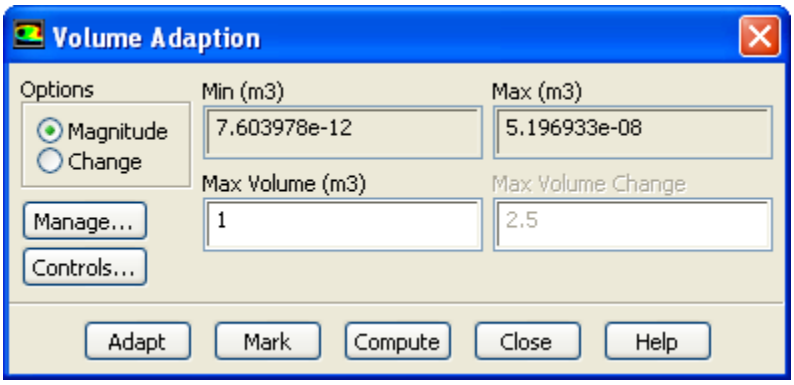
36.5.5. Adapt/Volume...

The **Adapt/Volume...** menu item opens the *Volume Adaption Dialog Box* (p. 2124).

36.5.5.1. Volume Adaption Dialog Box

The **Volume Adaption** dialog box allows you to mark or refine cells based on cell volume or change in cell volume. See *Volume Adaption* (p. 1337) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the radio buttons that toggle between marking and/or refining based on volume magnitude or volume change.

Magnitude

enables the marking/refining of cells based on volume magnitude.

Change

enables the marking/refining of cells based on the change in volume.

Min

displays the minimum value of cell volume or cell volume change in the mesh. This value is not editable.

Max

displays the maximum value of cell volume or cell volume change in the mesh. This value is not editable.

Max Volume

defines the threshold value for marking/refining the mesh based on volume magnitude. Cells that have volumes greater than the threshold are marked for refinement.

Max Volume Change

defines the threshold value for marking/refining the mesh based on the change in volume. Cells with volume changes that are greater than the threshold value are marked for refinement.

Manage...

opens the *Manage Adaption Registers Dialog Box* (p. 2129), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...

opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Adapt

refines the mesh based on either the maximum volume or the volume change, and the adaption limits.

Mark

marks cells to be refined based on either the maximum volume or the volume change. This command produces an adaption register. For information on using adaption registers, see *Manipulating Adaption Registers* (p. 1343).

Compute

calculates the minimum and maximum cell volume or cell volume change and displays them in the **Min** and **Max** real number fields.

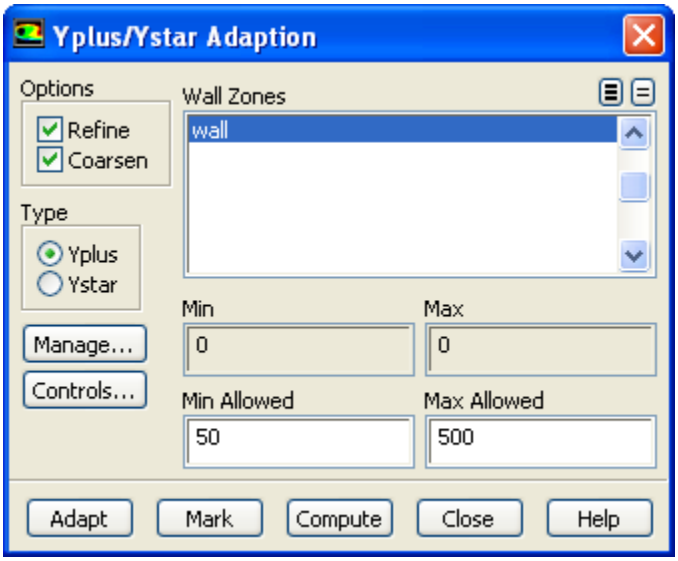
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.6. Adapt/Yplus/Ystar...

The **Adapt/Yplus/Ystar...** menu item opens the *Yplus/Ystar Adaption Dialog Box* (p. 2126).

36.5.6.1. Yplus/Ystar Adaption Dialog Box

The **Yplus/Ystar Adaption** dialog box allows you to mark or adapt boundary cells on specified wall zones based on the non-dimensional y^+ or y^* parameter. See *Yplus/Ystar Adaption* (p. 1338) for details.



Controls

Options

contains the check buttons that toggle the ability to mark and/or adapt cells for refinement or coarsening.

Refine

toggles the ability to refine cells or mark cells for refinement.

Coarsen

toggles the ability to coarsen cells or mark cells for coarsening.

Type

contains the check buttons that enable adaption based on y^+ or y^*.

Yplus

enables y^+ adaption.

Ystar

enables y^* adaption.

Wall Zones

contains a selectable list of active wall zones. Boundary cells associated with the wall zones you select will be marked or adapted based on the options, thresholds, and limitations applied in the dialog box.

Min/Max

displays the minimum and maximum cell values of y^+ or y^* for all cells associated with viscous wall zones. Note that these values are independent of the wall zones selected. The real number field values are not editable; they are purely informational.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min Allowed

designates the threshold value for coarsening the mesh. Cells with y^+ or y^* values below the minimum threshold will be marked for coarsening.

Max Allowed

designates the threshold value for refining the mesh. Cells with y^+ or y^* values above the maximum threshold will be marked for refinement.

Manage...

opens the *Manage Adaption Registers Dialog Box* (p. 2129), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...

opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Adapt

adapts the mesh based the coarsening and refining toggle buttons, the wall zones selected, the minimum and maximum y^+ or y^* allowed, and the adaption limits.

Mark

marks cells to be refined and/or coarsened based on the wall zones selected and the minimum and maximum y^+ or y^* allowed in the mesh. This command produces an adaption register. For information on using adaption registers, see *Manipulating Adaption Registers* (p. 1343).

Compute

computes the minimum and maximum values of y^+ or y^* on all cells on viscous walls and displays them in the **Min** and **Max** real number fields. Note that these are the extremes for the y^+ or y^* values of every cell on a viscous wall, not just the cells associated with the selected zones.

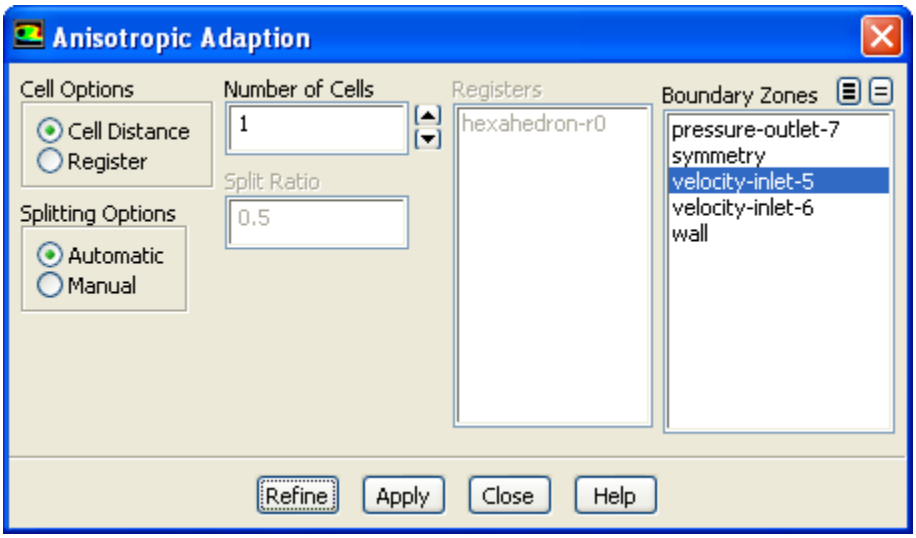
36.5.7. Adapt/Anisotropic...

The **Adapt/Anisotropic...** menu item opens the *Anisotropic Adaption Dialog Box* (p. 2127).

36.5.7.1. Anisotropic Adaption Dialog Box

The **Anisotropic Adaption** dialog box allows you to perform anisotropic adaption for certain cell types. See *Anisotropic Adaption* (p. 1339) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Cell Options
controls the marking of boundary layer cells.

Cell Distance
indicates adaption using the distance of the marked cell to the boundary zone.

Register
indicates adaption using an existing register.

Splitting Options
control how the splitting ratio is computed.

Automatic
allows the ratio to be computed automatically from the mesh.

Manual
allows you to enter a specific value for the ratio in the **Split Ratio** field.

Number of Cells
indicates the number of cells to be adapted and marked (available only when the **Cell Distance** option is enabled).

Split Ratio
allows you to specify a value for the split ratio (available only when the **Manual** splitting option is enabled).

Registers
provides a list of existing registers (available only when the **Register** option is enabled).

Boundary Zones
contains a list of available boundary zones.

Refine
refines the marked cells.

36.5.8. Adapt/Manage...

The **Adapt/Manage...** menu item opens the *Manage Adaption Registers Dialog Box* (p. 2129).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.8.1. Manage Adaption Registers Dialog Box

The **Manage Adaption Registers** dialog box provides an interactive mechanism for creating, destroying, and displaying functions for mesh adaption. See *Manipulating Adaption Registers* (p. 1343) for details about using the items in this dialog box.

Controls

Register Actions

contains operations applied to adaption or mask registers.

Change Type

toggles the register between adaption and mask types. Adaption registers are used to initiate refining or coarsening of the mesh. Typically, mask registers are combined with adaption registers to control the scope of the adaption process.

Combine

combines the selected adaption registers to create a hybrid adaption function. In some instances, three new registers may be created: a combination of the adaption registers, a combination of the mask registers, and then a combination of the two combined registers.

Delete

permanently discards the selected registers.

Mark Actions

contains operations applied to the cell markings defined in an adaption or mask register.

Exchange

modifies the cell markings in the following manner: all cells originally marked for refinement are marked for coarsening, and all cells originally marked for coarsening are marked for refinement.

Invert

modifies all the cell markings in a mask register in the following manner: all cells that were originally marked as ACTIVE are marked INACTIVE, and all cells originally marked as INACTIVE are marked ACTIVE. Note that this action can only be applied to mask registers.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limit
applies the adaption volume limits to the selected registers. For information on adaption limits, see *Mesh Adaption Controls* (p. 1347).

Fill
marks for coarsening all cells in the adaption register that are not marked for refinement.

Registers
contains a list from which you can select the current adaption and mask registers. Many of the actions in the dialog box are activated or deactivated based on the number and/or type of adaption registers selected.

Register Info
provides the name, ID, number of cells marked for refinement and coarsening, and the type of the most recently selected or deselected register.

Options...
opens the *Adaption Display Options Dialog Box* (p. 2134).

Controls...
opens the *Mesh Adaption Controls Dialog Box* (p. 2130), which allows you to control certain aspects of the adaption process.

Adapt
adapts the mesh based on the selected adaption register. The **Adapt** button is deactivated if more than one adaption register is selected. Adaption functions composed of combinations of adaption registers can be produced using commands in the **Register Actions** and **Mark Actions** boxes.

Display
displays the cells marked for adaption in the selected adaption register in the active graphics window. The **Display** button is deactivated if more than one adaption register is selected. Adaption functions composed of combinations of adaption registers can be produced using commands in the **Register Actions** and **Mark Actions** boxes.

36.5.9. Adapt/Controls...

The **Adapt/Controls...** menu item opens the *Mesh Adaption Controls Dialog Box* (p. 2130).

36.5.9.1. Mesh Adaption Controls Dialog Box

The **Mesh Adaption Controls** dialog box allows you to set limits on the minimum cell size, the minimum and maximum number of cells, and the cell types that can be adapted. In addition, it allows you to vary the volume weighting of the gradient adaption function. See *Mesh Adaption Controls* (p. 1347) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Options
contains check buttons that control the manner in which the mesh can be adapted.

Refine
toggles mesh adaption by adding points.

Coarsen
toggles mesh adaption by removing points.

Zones
contains a list of cell zones from which you can select the zones in which to perform adaption (or marking). By default, all cell zones are selected.

Min Cell Volume
restricts the size of the cell that is considered for refinement. Even if the cell is marked for refinement, it will not be refined if its cell volume is less than this threshold value.

Min # of Cells
specifies the minimum number of cells required in the mesh.

Max # of Cells
limits the total number of cells allowed in the mesh. A value of zero places no limits on the number of cells.

Max Level of Refine
specifies the maximum level of refinement for the cells.

Volume Weight
controls the volume weighting in the gradient adaption function. Valid values are between 0 and 1: 0 for no volume weighting and 1 for full volume weighting.

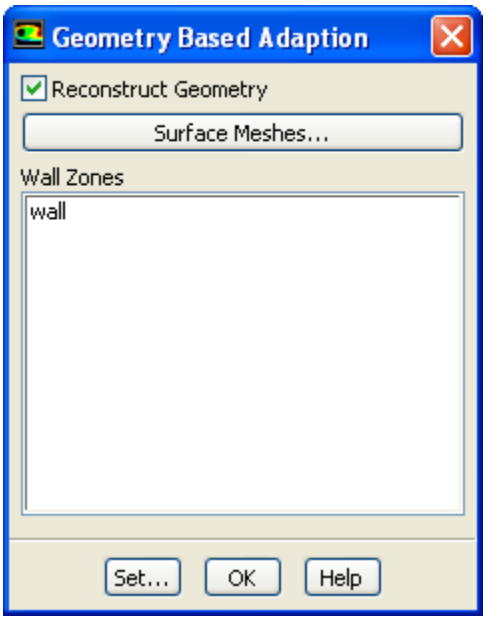
36.5.10. Adapt/Geometry...

The **Adapt/Geometry...** menu item opens the *Geometry Based Adaption Dialog Box* (p. 2132).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.10.1. Geometry Based Adaption Dialog Box

The **Geometry Based Adaption** dialog box allows you to reconstruct the geometry while performing boundary adaption. See *Geometry-Based Adaption* (p. 1341) for details.



Controls

Reconstruct Geometry
enables geometry based adaption parameters.

Surface Meshes...
opens the *Surface Meshes Dialog Box* (p. 2132).

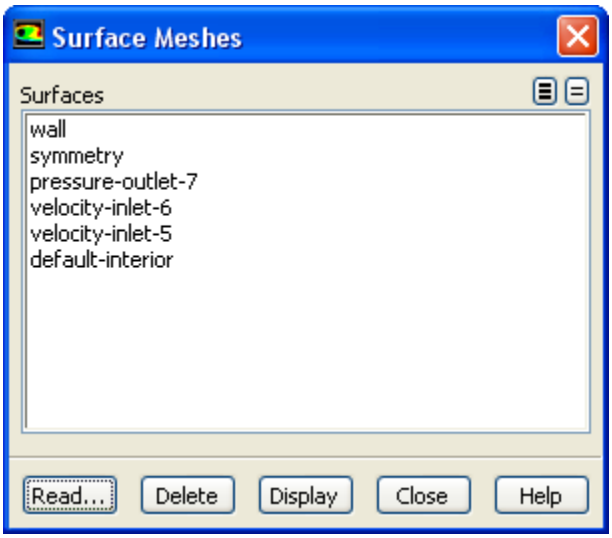
Wall Zones
allows you to select the wall zones you want to adapt.

Set...
opens the *Geometry Based Adaption Controls Dialog Box* (p. 2133).

36.5.10.2. Surface Meshes Dialog Box

The **Surface Meshes** dialog box allows you to read the surface meshes in ANSYS FLUENT (See *Reading Surface Mesh Files* (p. 160) for details on reading surface meshes.)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Surfaces
contains a list of the surfaces available in the surface mesh you read.

*You can select/deselect the surfaces listed under **Surfaces**.*

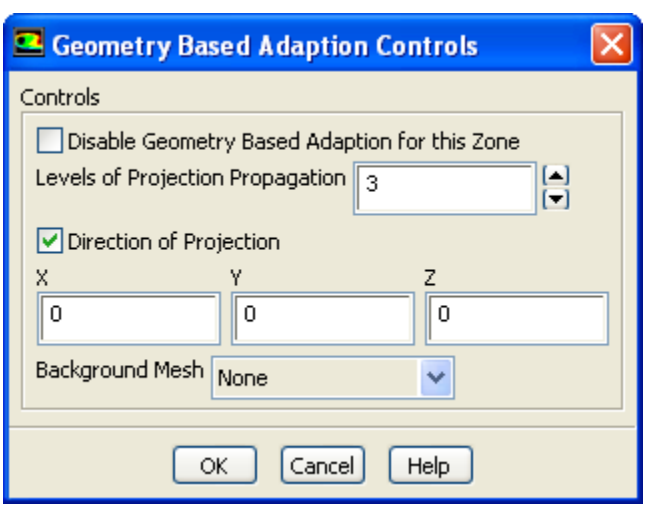
Read...
opens the **Select File** dialog box in which you can select the surface mesh you want to read.

Delete
allows you to delete the selected surfaces under **Surfaces** area.

Display
allows you to display the selected surfaces under **Surfaces**.

36.5.10.3. Geometry Based Adaption Controls Dialog Box

The **Geometry Based Adaption Controls** dialog box allows you to set projections. It is opened from the *Geometry Based Adaption Dialog Box* (p. 2132). See *Performing Geometry-Based Adaption* (p. 1341) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

contains parameters to define geometry based adaption.

Disable Geometry Based Adaption for this Zone

disables the geometry reconstruction for selected zones in the domain.

Levels of Projection Propagation

is the number of layers of the nodes you want to project.

Direction of Projection

allows you to specify the direction in which you want to project the nodes.

Background Mesh

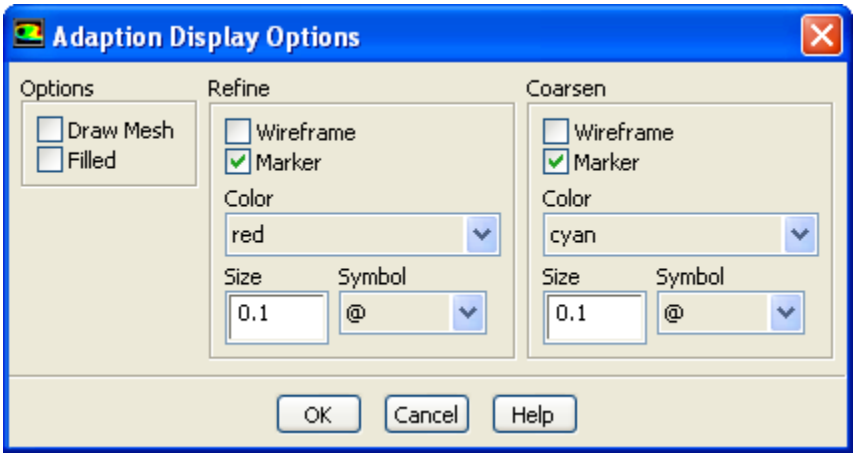
allows you to load the surface mesh as a background mesh for adaption.

36.5.11. Adapt/Display Options...

The **Adapt/Display Options...** menu item opens the *Adaption Display Options Dialog Box* (p. 2134).

36.5.11.1. Adaption Display Options Dialog Box

The **Adaption Display Options** dialog box allows you to customize the display of adaption or mask registers. See *Adaption Display Options* (p. 1346) for details about the items below.



Controls

Options

contains check buttons that control the drawing of the mesh and the type of graphical tool used to display flagged cells.

Draw Mesh

toggles the ability to draw the mesh with the adaption display. This command opens the *Mesh Display Dialog Box* (p. 1635), which allows you to select the desired surface or zone meshes to be displayed with the markings.

Filled

toggles the solid shading of the cell wireframe.

Refine

contains options related to the display of cells marked for refinement.

Wireframe

toggles the display of the cell wireframe for cells flagged for refinement.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Marker
toggles the display of the cell marker for cells flagged for refinement.

Color
is a drop-down list of colors for the wireframe or marker for the cells marked for refinement.

Size
is a real number entry for the size of the refine cell marker. A symbol of size 1.0 is 3.0% of the height of the display screen.

Symbol
is a drop-down list of symbols that can be used for the refine cell marker.

Coarsen
contains options related to the display of cells marked for refinement.

Wireframe
toggles the display of the cell wireframe for cells flagged for coarsening.

Marker
toggles the display of the cell marker for cells flagged for coarsening.

Color
is a drop-down list of colors for the wireframe or marker for the cells marked for coarsening.

Size
is a real number entry for the size of the coarsen cell marker. A symbol of size 1.0 is 3.0% of the height of the display screen.

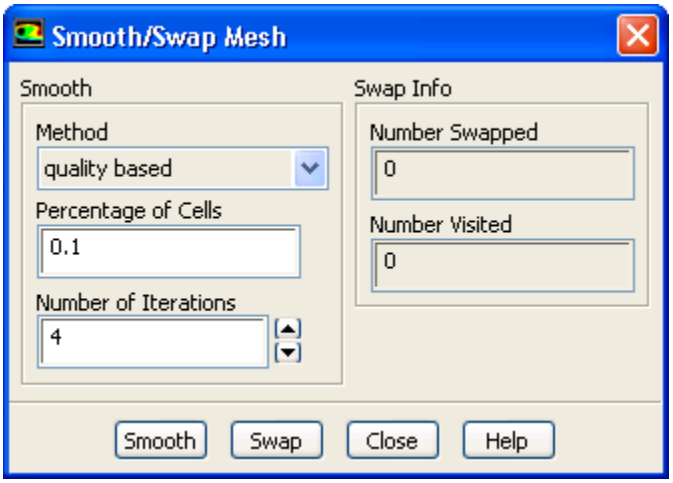
Symbol
is a drop-down list of symbols that can be used for the coarsen cell marker.

36.5.12. Adapt/Smooth/Swap...

The **Adapt/Smooth/Swap...** menu item opens the *Smooth/Swap Mesh Dialog Box* (p. 2135).

36.5.12.1. Smooth/Swap Mesh Dialog Box

The **Smooth/Swap Mesh** dialog box controls smoothing and face swapping of the numerical mesh. See *Improving the Mesh by Smoothing and Swapping* (p. 1349) for details about the items below.



Controls

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smooth

contains parameters associated with smoothing the mesh.

Method

indicates whether the quality-based, Laplacian, or the skewness-based smoothing method is to be used. Select **quality based**, **laplace**, or **skewness** from the drop-down list. Note that only the **quality based** item is available for parallel cases.

Percentage of Cells

(for quality-based smoothing) sets the percentage of the total cells on which improvements are attempted. Note that the cells selected for improvement are in the areas of the mesh that exhibit the lowest orthogonal quality. This field appears when **quality based** is selected as the smoothing method.

Relaxation Factor

(for Laplacian smoothing) sets the factor by which to multiply the computed position increment for the node. The lower the factor, the more reduction in node movement. This field appears when **laplace** is selected as the smoothing method.

Skewness Threshold

(for skewness-based smoothing) sets the minimum cell skewness value for which node smoothing will be attempted. ANSYS FLUENT will try to move interior nodes to improve the skewness of cells with skewness greater than this value. By default, **Skewness Threshold** is set to 0.8 for 3D or 0.4 for 2D. The **Skewness Threshold** field appears when **skewness** is selected as the smoothing method.

Number of Iterations

defines the number of successive smoothing sweeps performed on the mesh.

Swap Info

provides information on the most recent face-swapping operation. This group box is only available for serial cases.

Number Swapped

displays the number of faces that were swapped based on the Delaunay circle test.

Number Visited

displays the total number of faces that were visited and tested for possible face swapping.

Smooth

initiates the desired number of smoothing iterations.

Swap

exchanges the faces of cells for which the circle test is not satisfied.

36.6. Surface Menu

For additional information, please see the following sections:

36.6.1. Surface/Zone...
36.6.2. Surface/Partition...
36.6.3. Surface/Point...
36.6.4. Surface/Line/Rake...
36.6.5. Surface/Plane...
36.6.6. Surface/Quadric...
36.6.7. Surface/Iso-Surface...
36.6.8. Surface/Iso-Clip...
36.6.9. Surface/Transform...
36.6.10. Surface/Manage...

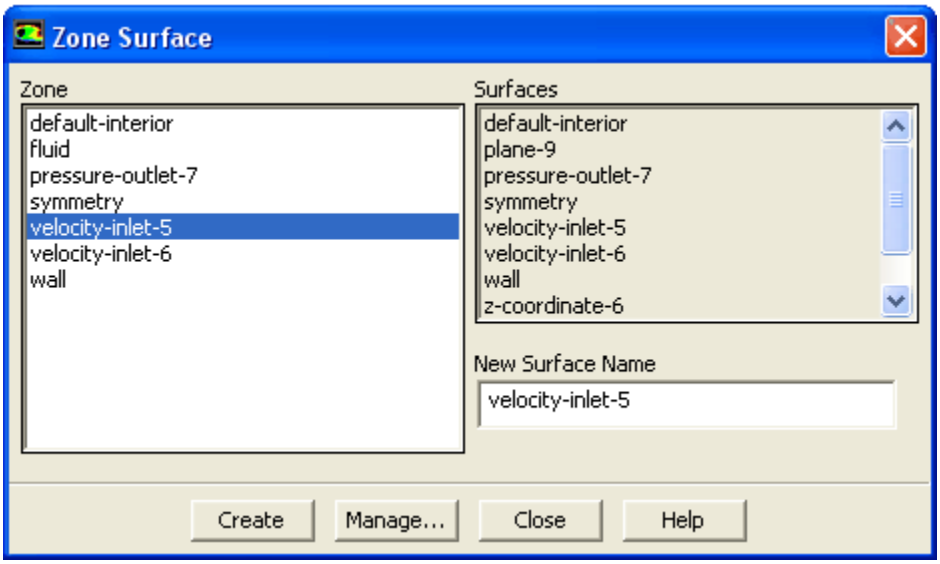
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.6.1. Surface/Zone...

The **Surface/Zone...** menu item opens the *Zone Surface Dialog Box* (p. 2137).

36.6.1.1. Zone Surface Dialog Box

The **Zone Surface** dialog box allows you to interactively create surfaces from face and cell zones in the domain. See *Zone Surfaces* (p. 1356) for details about the items below.



Controls

Zone
contains a list from which you can select the face or cell zone to be used for creating a zone surface.

Surfaces
displays an informational list of existing surfaces.

New Surface Name
designates the name of the surface to be created. The default is the zone name.

Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

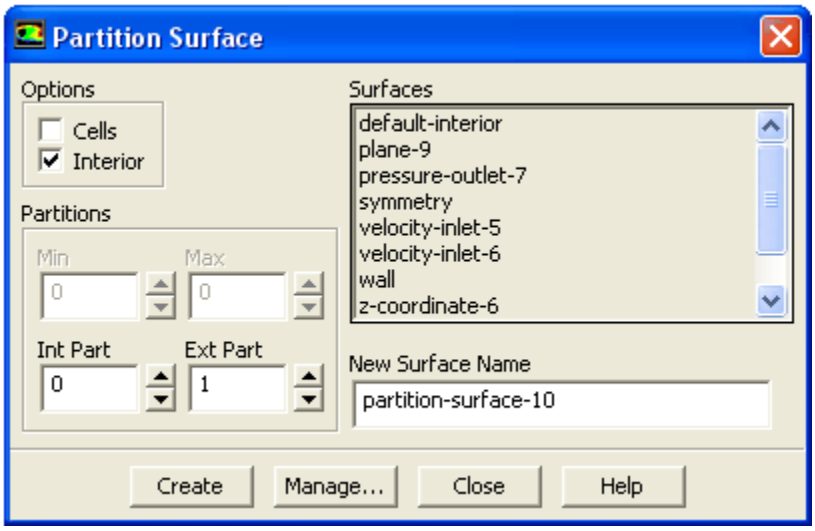
36.6.2. Surface/Partition...

The **Surface/Partition...** menu item opens the *Partition Surface Dialog Box* (p. 2137).

36.6.2.1. Partition Surface Dialog Box

The **Partition Surface** dialog box allows you to create a surface defined by the boundary of two adjacent mesh partitions. See *Partition Surfaces* (p. 1357) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains check buttons that allow you to choose interior or exterior faces or cells to be contained in the partition surface.

Cells

specifies, when enabled, that the partition surface will consist of cells (either interior or exterior, depending on the status of the **Interior** check button) that lie on the partition boundary. When this option is disabled, the faces on the boundary between partitions will make up the partition surface. Depending on the status of the **Interior** check button, the faces will reflect data values for the interior or exterior cells.

Interior

toggles between interior and exterior cells. If the partition surface consists of faces instead of cells, **Interior** specifies for which cells data will be displayed on the faces. When this option is enabled, interior cells (cells that are on the **Int Part** side of the partition boundary) will make up the partition surface. When it is disabled, the partition surface will consist of exterior cells (cells that are on the **Ext Part** side of the partition boundary).

Partitions

contains information about which partitions are under consideration. The boundary that defines the partition surface is the boundary between the **Int Part** and the **Ext Part**. If there are more than two mesh partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for **Int Part** and **Ext Part**, you can create surfaces for any of these boundaries.

Min/Max

indicates the minimum and maximum ID numbers of the mesh partitions. The minimum is always zero, and the maximum is one less than the number of processors.

Int Part

indicates the ID number of the interior partition (i.e., the par tition under consideration).

Ext Part

indicates the ID number of the bordering partition.

Surfaces

displays an informational list of existing surfaces.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

36.6.3. Surface/Point...

The **Surface/Point...** menu item opens the *Point Surface Dialog Box* (p. 1927).

36.6.4. Surface/Line/Rake...

The **Surface/Line/Rake...** menu item opens the *Line/Rake Surface Dialog Box* (p. 1928).

36.6.5. Surface/Plane...

The **Surface/Plane...** menu item opens the *Plane Surface Dialog Box* (p. 1929).

36.6.6. Surface/Quadric...

The **Surface/Quadric...** menu item opens the *Quadric Surface Dialog Box* (p. 1931).

36.6.7. Surface/Iso-Surface...

The **Surface/Iso-Surface...** menu item opens the *Iso-Surface Dialog Box* (p. 1933).

36.6.8. Surface/Iso-Clip...

The **Surface/Iso-Clip...** menu item opens the *Iso-Clip Dialog Box* (p. 1935).

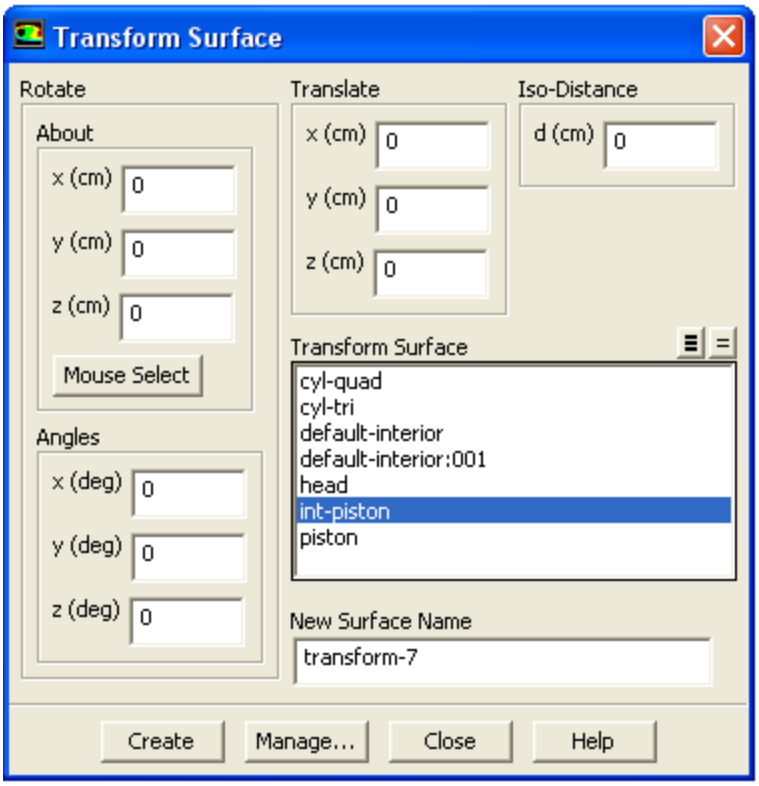
36.6.9. Surface/Transform...

The **Surface/Transform...** menu item opens the *Transform Surface Dialog Box* (p. 2139).

36.6.9.1. Transform Surface Dialog Box

The **Transform Surface** dialog box allows you to create a new data surface by rotating and/or translating an existing surface, and/or by specifying a constant normal distance from it. See *Transforming Surfaces* (p. 1375) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Rotate

contains the transformation parameters for rotation.

About

defines the origin about which the surface is rotated. You will specify a point, and the origin of the coordinate system for the rotation will be set to the specified point. For example, if you specified the point (1,0) in 2D, rotation would be about the z axis anchored at (1,0). You can either enter the point's coordinates in the **x,y,z** fields or click on the **Mouse Select** button and select a point in the graphics window using the mouse.

Angles

define the angles about the **x**, **y**, and **z** axes (i.e., the axes of the coordinate system with the origin defined under **About**) by which the surface is rotated. For 2D problems, you can specify rotation about the **z** axis only.

Translate

contains the transformation parameters for translation.

x,y,z

define the distance by which the surface is translated in each direction.

Iso-Distance

contains the transformation parameters for "isodistancing."

d

sets the normal distance between the original surface and the transformed surface.

Transform Surface

contains a list of existing surfaces from which you can select the surface to be transformed. The selected surface will remain unchanged; the transformation will create a new surface.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Surface Name
designates the name of the new surface. The default is the concatenation of the transformation type (i.e., iso-distance, rotate, or translate) and an integer which is the new surface ID.

Create
creates the surface.

Manage...
opens the *Surfaces Dialog Box* (p. 1936) in which you can rename and delete surfaces and determine their sizes.

36.6.10. Surface/Manage...

The **Surface/Manage...** menu item opens the *Surfaces Dialog Box* (p. 1936).

36.7. Display Menu

For additional information, please see the following sections:

36.7.1. Display/Mesh...
36.7.2. Display/Graphics and Animations...
36.7.3. Display/Plots...
36.7.4. Display/Residuals...
36.7.5. Display/Options...
36.7.6. Display/Scene...
36.7.7. Display/Views...
36.7.8. Display/Lights...
36.7.9. Display/Colormap...
36.7.10. Display/Annotate...
36.7.11. Display/Zone Motion...
36.7.12. Display/DTRM Graphics...
36.7.13. Display/Import Particle Data...
36.7.14. Display/PDF Tables/Curves...
36.7.15. Display/Video Control...
36.7.16. Display/Mouse Buttons...

36.7.1. Display/Mesh...

The **Display/Mesh...** menu item opens the *Mesh Display Dialog Box* (p. 1635).

36.7.2. Display/Graphics and Animations...

The **Display/Graphics and Animations...** menu item opens the *Graphics and Animations Task Page* (p. 1965).

36.7.3. Display/Plots...

The **Display/Plots...** menu item opens the *Plots Task Page* (p. 2011).

36.7.4. Display/Residuals...

The **Display/Residuals...** menu item opens the *Residual Monitors Dialog Box* (p. 1915).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.7.5. Display/Options...

The **Display/Options...** menu item opens the *Display Options Dialog Box* (p. 1992).

36.7.6. Display/Scene...

The **Display/Scene...** menu item opens the *Scene Description Dialog Box* (p. 1994). This menu item becomes available when you display meshes, surfaces, contours, vectors, or pathlines in the graphics window.

36.7.7. Display/Views...

The **Display/Views...** menu item opens the *Views Dialog Box* (p. 2000).

36.7.8. Display/Lights...

The **Display/Lights...** menu item opens the *Lights Dialog Box* (p. 2005).

36.7.9. Display/Colormap...

The **Display/Colormap...** menu item opens the *Colormap Dialog Box* (p. 2007).

36.7.10. Display/Annotate...

The **Display/Annotate...** menu item opens the *Annotate Dialog Box* (p. 2010).

36.7.11. Display/Zone Motion...

The **Display/Zone Motion...** menu item opens the *Zone Motion Dialog Box* (p. 1895). This option is available for all dynamic mesh models.

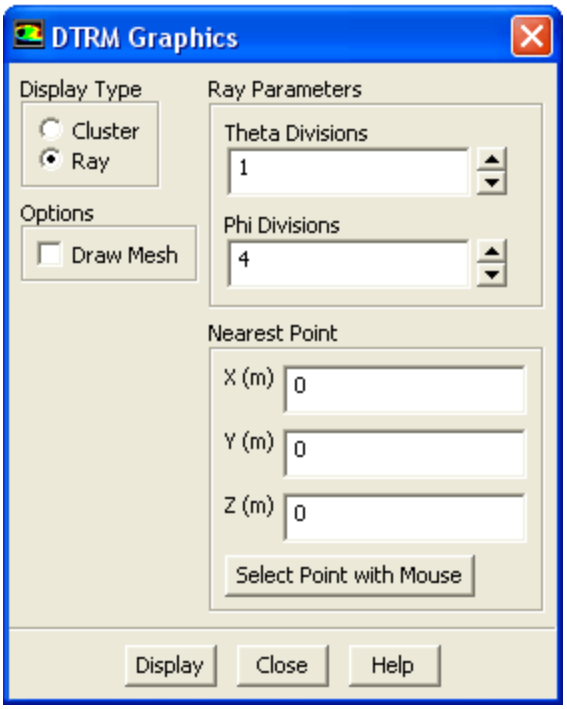
36.7.12. Display/DTRM Graphics...

The **Display/DTRM Graphics...** menu item opens the *DTRM Graphics Dialog Box* (p. 2142).

36.7.12.1. DTRM Graphics Dialog Box

The **DTRM Graphics** dialog box allows you to display rays and clusters used by the DTRM. See *Displaying Rays and Clusters for the DTRM* (p. 738) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Display Type
contains options for the different items you can display.

Cluster
specifies the display of clusters.

Ray
specifies the display of rays.

Options
contains check buttons that control display options.

Draw Mesh
toggles between displaying and not displaying the mesh. The *Mesh Display Dialog Box* (p. 1635) is opened when **Draw Mesh** is selected.

Cluster Type
specifies whether **Surface** clusters or **Volume** clusters are to be displayed. This section of the dialog box appears when **Cluster** is selected as the **Display Type**.

Cluster Selection
contains options for cluster displays. This section of the dialog box appears when **Cluster** is selected as the **Display Type**.

Display All Clusters
enables the display of all surface or volume clusters in the domain.

Ray Parameters
contains controls for displaying rays. This section of the dialog box appears when **Ray** is selected as the **Display Type**.

Theta Divisions, Phi Divisions
control the number of rays being traced. (See *Controlling the Rays* (p. 708).)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nearest Point
specifies a point (**X,Y,Z**) near the cluster to be displayed (or the cluster from which the rays should start).

The **Nearest Point** controls are not available when **Display All Clusters** is selected under **Cluster Selection**.

Select Point With Mouse
is an alternative method for specifying the **Nearest Point** using your mouse, by clicking the button.

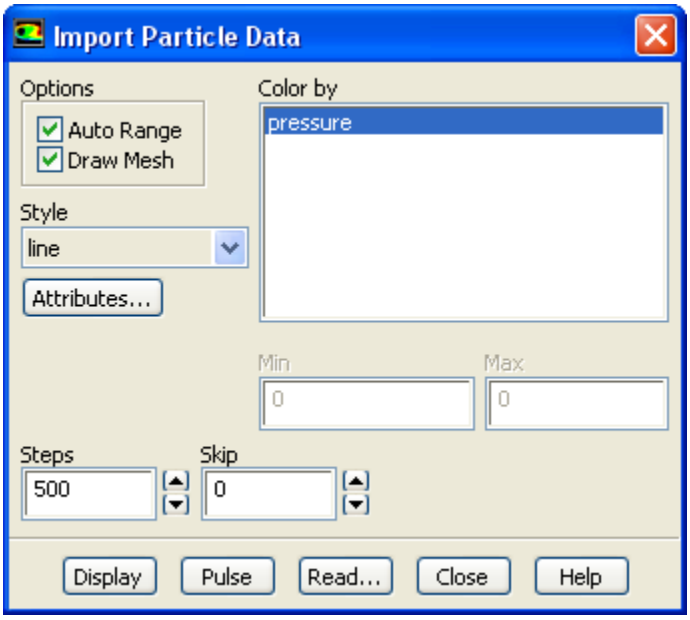
Display
displays the specified cluster(s) or rays.

36.7.13. Display/Import Particle Data...

The **Display/Import Particle Data...** menu item opens the *Import Particle Data Dialog Box* (p. 2144).

36.7.13.1. Import Particle Data Dialog Box

The **Import Particle Data** dialog box allows you to import particle history data for display purposes. See *Importing Particle Data* (p. 1070) for details.



Controls

Options
contains the check buttons that set various import particle data options.

Auto Range
toggles between automatic and manual setting of the particle data range.

Draw Mesh
toggles between displaying and not displaying the mesh. The *Mesh Display Dialog Box* (p. 1635) is opened when **Draw Mesh** is selected.

Style
allows you to select pathline style.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Attributes...
opens the *Path Style Attributes Dialog Box* (p. 1979). This allows you to modify the line width, cylinder radius or marker size.

Color by
contains a list from which you can select the scalar field to be used to color the particle data.

Min,Max
allows you to set the range when **Auto Range** is disabled.

Steps
sets the maximum number of steps a particle can advance.

Skip
allows you to "thin out" the pathlines.

Display
displays pathlines.

Pulse
animates the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

Read...
opens a file selection dialog box where you can enter a file name and a directory for the imported data.

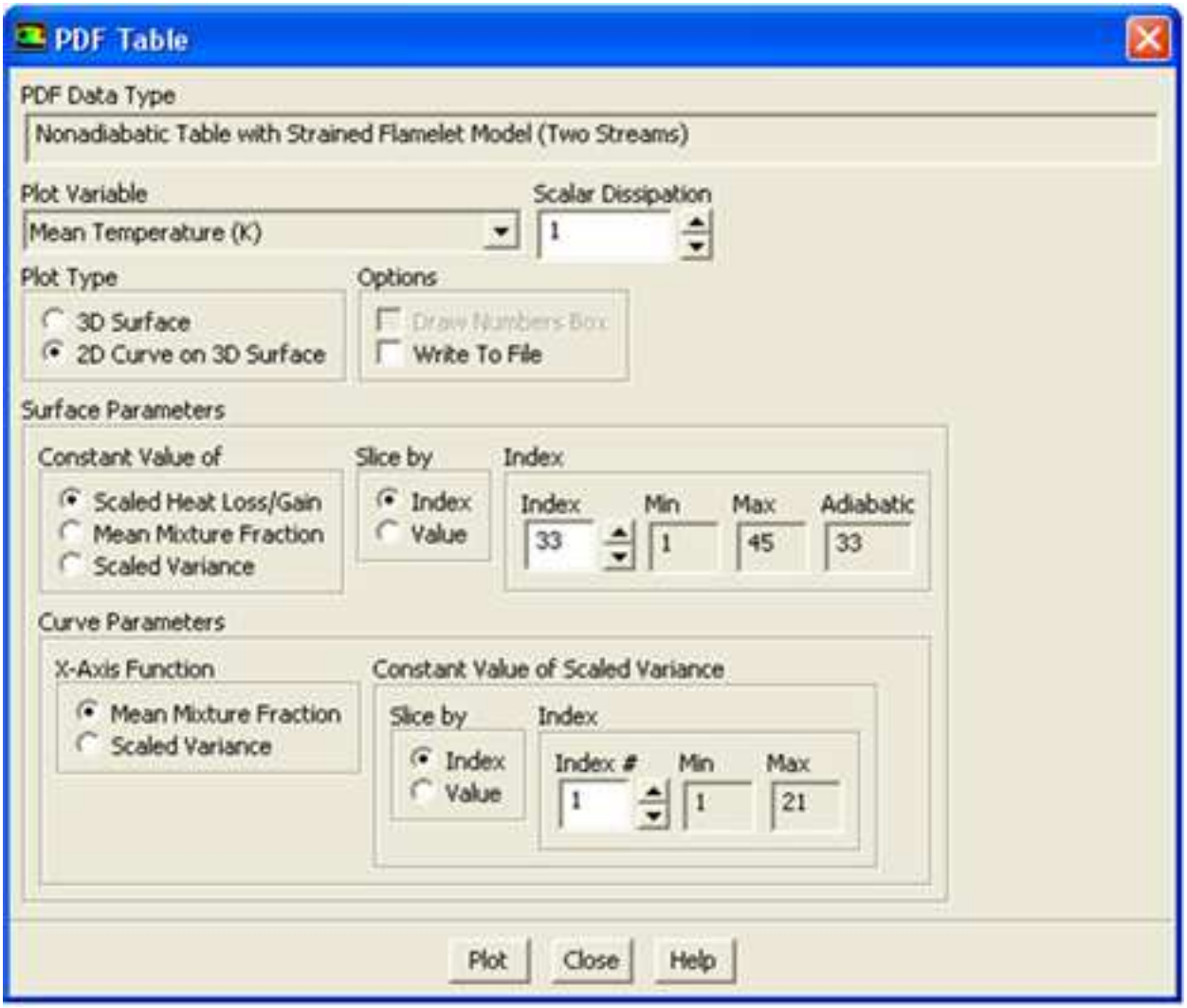
36.7.14. Display/PDF Tables/Curves...

The **Display/PDF Tables/Curves...** menu item opens the *PDF Table Dialog Box* (p. 2145).

36.7.14.1. PDF Table Dialog Box

You can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy. See *Postprocessing the Look-Up Table Data* (p. 874) for details about the items below.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

PDF Data Type
describes the system that you are displaying.

Plot Variable
contains a drop-down list from which you can select temperature, density, or species fraction as the variable to be plotted.

Scalar Dissipation
specifies the value of the **Scalar Dissipation** which is available for multiple flamelets only.

Plot Type
gives you the choice of plotting a 3D surface or a slice of a 3D surface.

3D Surface
displays a 3D plot of the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy.

2D Curve on 3D Surface
consists of a 2D curve which is a slice of a 3D surface.

Options
contains options specific to the display of 3D surfaces or 2D curves on 3D surfaces.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Draw Numbers Box
enabling this option displays a wireframe box with the numerical limits in each coordinate direction. This option is available only when **3D Surface** is selected.

Write To File
specifies whether you want to write the plot data to a file. This option is available only when **2D Curve on 3D Surface** is selected. The **Plot** button changes to a **Write** button when this option is enabled.

Surface Parameters
contains settings where discrete independent variables are held constant and where curve parameters are defined.

Constant Value of
specifies the discrete independent variable to be held constant in the lookup table. The choices are **Scaled Heat Loss/Gain**, **MeanMixtureFraction**,or **Scaled Variance**. For a two-mixture-fraction case, the **Scaled Heat Loss/Gain** is the only available option.

Slice by
allows you to select whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value**.

Index/Value
contains index/values and their ranges.

Index/Value
allows you to specify the discretization index or numerical value of the variable that is being held constant.

Min/Max
are the range of integer values that you are allowed to choose from or display.

Adiabatic
is the enthalpy slice index corresponding to the adiabatic case for which the enthalpy (**Scaled Heat Loss/Gain**) is held constant.

Curve Parameters
allows you to specify the **X-Axis Function** against which the plot variable will be displayed when **2D Curve on 3D Surface** is selected.

X-Axis Function
allows you to select **Mean Mixture Fraction** or **Scaled Variance** against which the plot variable will be displayed.

Constant Value of Scaled Variance/Mean Mixture Fraction
allows you to specify the type of discretization for the variable that is being held constant.

Slice by
allows you to select whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value**.

Index/Value
contains index/values and their ranges.

Index#/Value
allows you to specify the discretization index or numerical value of the variable that is being held constant for the curve parameters.

Min/Max
are the range of integer values that you are allowed to choose from or display.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display
displays the plot variable of the 3D surface.

Plot
plots the plot variable for the 2D curve on 3D surface.

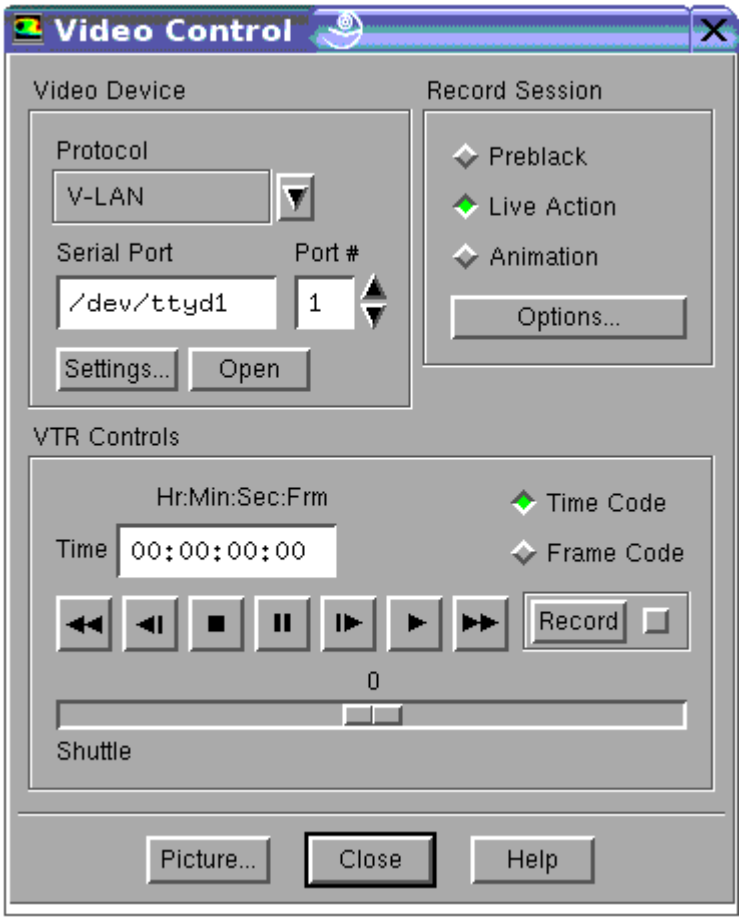
Write
opens the *The **Select File** Dialog Box* (p. 33) where you will specify a name for the file containing plot data. This button appears when **Write To File** is enabled for the **2D Curve on 3D Surface** plot type.

36.7.15. Display/Video Control...

The **Display/Video Control...** menu item opens the *Video Control Dialog Box* (p. 2148) (Linux/UNIX only).

36.7.15.1. Video Control Dialog Box

The **Video Control** dialog box is used to record animations or live action to video. When recording animations to video, you must first create your animation (see *Creating an Animation* (p. 1455)). The **Video Control** dialog box will play your animation and control a videotape recorder (VTR) to record it to video. See *Creating Videos* (p. 1458) for details about using the video tools.



Controls

Video Device
sets up the connection to the VTR control hardware.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Protocol

specifies the command protocol that identifies your VTR controller. ANSYS FLUENT currently supports the following VTR controllers:

V-LAN

Machine Control Network is a command protocol developed by Videomedia, Inc. and supported by several videotape editing systems.

MiniVAS

is the command protocol for the MiniVAS and MiniVAS-2 VTR controllers developed by the V.A.S. Group.

To be able to control the video recording from ANSYS FLUENT, your VTR controller must use one of these protocols.

Serial Port

identifies the name of the serial port to which your VTR controller is connected.

Port #

is a convenient way of changing the port number associated with the serial port.

Settings...

displays a dialog box that provides options specific to your VTR controller. If your VTR controller protocol is V-LAN, the *V-LAN Settings Dialog Box* (p. 2150) is displayed; if the protocol is MiniVAS, the *MiniVAS Settings Dialog Box* (p. 2152) is displayed.

Open

will open a connection to your VTR controller. Once the connection is open, the name of this push button will be changed to **Close** so that you can close the connection again.

Record Session

specifies the type of recording you want to perform.

Preblack

is the process of formatting a tape by laying down a time code onto the tape. A tape must be formatted before any frame accurate editing, including frame-by-frame animation, can be performed. During this process, one usually records a black video signal onto the tape as well, thus the name "preblack". When you select this option the current graphics window will be cleared to black. You can use the window to send your black video signal to the VTR. Remember, when you preblack a previously formatted tape, a new time code will be written and any previously recorded video will be destroyed.

Live Action

allows you to record a live ANSYS FLUENT session which can be used for demonstration. This option requires your computer's video hardware to have a scan converter that will send the computer display image to your VTR system.

Animation

will play an animation that you have created, and record it onto your VTR system.

Options...

displays the *Animation Recording Options Dialog Box* (p. 2153), which provides options for recording your animation.

VTR Controls

function like the controls on your VTR's front dialog box.

Time

is a counter that provides input/output of the current tape position. It will update automatically when the tape is repositioned. If you type a new time in this field and press `Enter`, the VTR will go

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to that position on the tape. For time code, the format is Hrs:Min:Sec:Frames, where the number of frames can range between 0 and 29. For example, a time code of `00:02:36:07` is 2 minutes, 36 seconds, and 7 frames. In order to go to this position on the tape, you can enter the time code as `2:36:07`, leaving out the leading zeros, or you can simply enter `23607`, leaving out the leading zeros and colons. If you select the **Frame Code** radio button, the tape position will be displayed in frames instead of time.

Record
will begin a recording session starting at the current position on the tape. If **Preblack** has been selected as the recording session, the label of this push button will be changed to **Preblack**.

Push Button Controls
enable you to operate the VTR's tape transport.

<<	rewinds the tape.
<|	reverses the tape by one frame.
[ ]	places the tape in stop mode.
||	places the tape in pause mode.
|>	advances the tape by one frame.
>	plays the tape at normal speed.
>>	advances the tape in fast-forward mode.

Shuttle
allows you to advance (or reverse) the tape at varying speeds. When you move the shuttle to the right, the tape will be advanced; when you move it to the left, the tape will be reversed. The shuttle speed index ranges from -9 to 9. A speed index of 5 is equivalent to playing the tape at normal speed. A speed index of 0 is equivalent placing the tape in pause mode.

Counter Format
allows you to change the counter units displayed in the **Time/Frame** counter.

Time Code
when selected, displays the tape position as time code.

Frame Code
when selected, displays the tape position as frame code.

This format is for display/user-input purposes only and does not change the actual tape counter format.

Picture...
displays the *Picture Options Dialog Box* (p. 2155), which provides options for setting the picture size and color levels in the graphics window.

36.7.15.2. V-LAN Settings Dialog Box

The **V-LAN Settings** dialog box allows you to set specific V-LAN options. This dialog box is displayed by clicking on the **Settings...** push button in the *Video Control Dialog Box* (p. 2148), after you have selected V-LAN as your video device protocol.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

V-LAN Node

identifies the node number of the VTR. This is always 1, unless you have several video devices connected to a V-LAN network, in which case, the VTR you want to use may be identified by another node number.

VTR Video Format

identifies the video format standard to which your VTR conforms. This setting is used to determine the video frame rate (frames/sec).

NTSC

(National Television Standards Committee) is the main video standard used in North America and Japan. The video frame rate is 30 frames/sec.

PAL

(Phase Alternate Line) is the video standard used by some countries in Western Europe, Asia, and Australia. The video frame rate is 25 frames/sec.

Preroll/Postroll

specifies the preroll and postroll times in seconds. These are used when performing **Edit** records, including frame-by-frame animation recording, to make sure the tape is moving at the proper speed through the recorded segment.

Preroll

is the number of seconds the tape will play up to the record in point.

Postroll

is the number of seconds the tape will play past the record out point.

The default settings should be sufficient for most VTR devices.

Live/Real-Time Recording

specifies the type of recording that will be performed during a live action or real-time animation recording session.

Quick Record

can be performed on a tape that has not been formatted or "preblacked". This is also known as "full record". Remember that this option will destroy any pre-existing time code on the tape.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit (Preblacked Tape Required)
is used for more precise recording. This mode allows you to record several back-to-back recordings with clean transitions between them.

For frame-by-frame animation recording, the record mode used is **Edit** regardless of this setting.

Verbose Mode
enables the echoing of each V-LAN command as it is sent to the V-LAN controller.

36.7.15.3. MiniVAS Settings Dialog Box

The **MiniVAS Settings** dialog box allows you to set specific MiniVAS options. This dialog box is displayed by clicking on the **Settings...** push button in the *Video Control Dialog Box* (p. 2148), after you have selected MiniVAS as your video device protocol.

Controls

VTR Identification
identifies the type of VTR the MiniVAS is connected to. The identification is a two-character code reserved for your VTR model (see your MiniVAS documentation for details). If left blank, this field is ignored and the MiniVAS will use the code that has been permanently stored in its ROM. It is recommended that you leave this field blank unless you know that a code change is required. If you open this dialog box after opening a connection to the MiniVAS, this field will contain the code that is stored in the MiniVAS.

VTR Video Format
identifies the video format standard to which your VTR conforms. This setting is used to determine the video frame rate (frames/sec).

NTSC
(National Television Standards Committee) is the main video format used in North America and Japan. The video frame rate is 30 frames/sec.

PAL
(Phase Alternate Line) is the video format used by some countries in Western Europe, Asia, and Australia. The video frame rate is 25 frames/sec.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VTR Tape Format
specifies which counter format to use for editing. MiniVAS provides commands based on both time code and frame code.

Time Code
instructs MiniVAS to control the VTR using time code.

Frame Code
instructs MiniVAS to control the VTR using frame code.

Preroll
specifies the preroll time in seconds. This is used when performing **Edit** records, including frame-by-frame animation recording, to make sure the tape is moving at the proper speed through the recorded segment. The value given is the number of seconds the tape will play up to the record in point. The default setting should be sufficient for most VTR devices.

Preblack Mode
specifies the mode used when you perform a preblack recording session.

Preblack
instructs the MiniVAS to perform a "hard" record on a previously unformatted tape. During this operation, the MiniVAS will command the VTR to write both a time code and a VIFC frame code to the tape.

Prestripe
instructs the MiniVAS to perform an "edit" record on a previously formatted tape that already has a time code. During this operation, the MiniVAS will command the VTR to write a VIFC frame code to the tape.

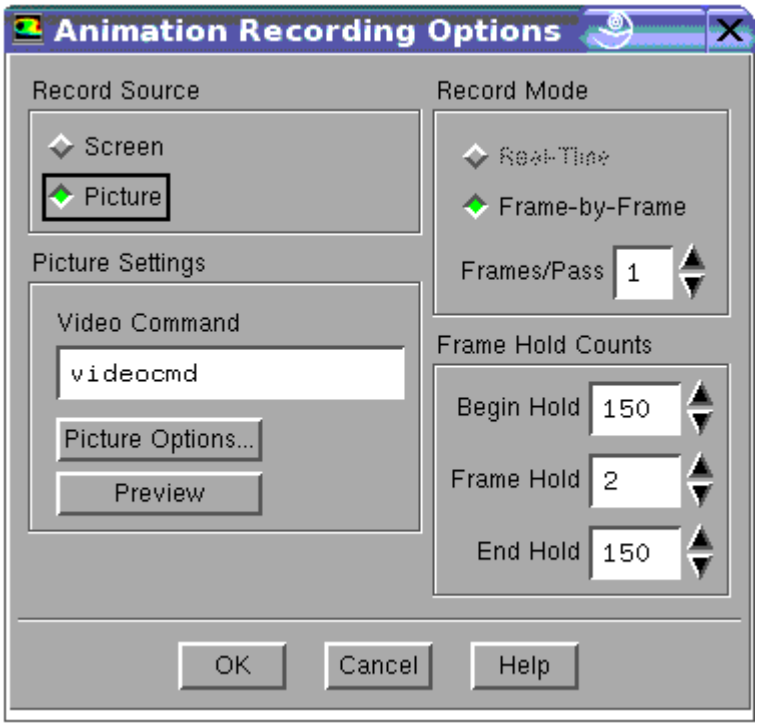
In either case, any previously recorded video will be destroyed.

Verbose Mode
enables the echoing of each MiniVAS command as it is sent to the MiniVAS controller.

36.7.15.4. Animation Recording Options Dialog Box

The **Animation Recording Options** dialog box allows you to set specific options for recording animations to video. An animation is created using the *Animate Dialog Box* (p. 1986).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Record Source

identifies the video source used during a recording session.

Screen

informs ANSYS FLUENT that the computer's video hardware will send all or a portion of the computer screen as a video signal to the VTR. This option assumes your computer's video system includes a scan converter and associated software that converts the computer RGB signal to a video signal. You are responsible for setting up the scan converter and sending the video signal to the VTR.

Picture

instructs ANSYS FLUENT to create a picture of each frame of animation and send the picture file to the computer's video hardware using a system command. This option assumes that your computer's video system includes a frame buffer that can store an image and send it as a video signal to the video recording system.

Picture Settings

is used in conjunction with the **Picture** record source option.

Video Command

identifies the shell script command file used to send a picture file to the video frame buffer. The default setting is `videocmd`, which is a shell script that is included in your ANSYS FLUENT distribution. It is located in *path* `/Fluent.Inc/bin`, where *path* is the directory in which you have placed the release directory, `Fluent.Inc`. This shell script will execute your system's command to send an image file to the video frame buffer. The script `videocmd` is set up to call the SGI system command `memtovid`. If you have a different system, you must copy the shell script `videocmd` to a new file and modify it to perform the proper task on your system (see the comments in `videocmd` for details).

Picture Options...

displays the *Save Picture Dialog Box* (p. 1988), which allows you to set up the picture format supported by your video frame buffer. If you choose to perform a window dump to create the picture file, the default window dump command used will also be `videocmd`. You can change this setting to use

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

your own command. After setting the picture options, click on **Apply** instead of **Save...**in the *Save Picture Dialog Box* (p. 1988) to effect the change.

Preview
will send a test picture of the current graphics picture to the video frame buffer. This can be used to check the image on your video recording system's monitor to see if the image size and color levels are acceptable.

Record Mode
specifies the method used to record the animation.

Real-Time
can be used if the animation playback speed is fast enough to provide a reasonably smooth animation in real-time. This is only available if the selected record source is **Screen**. In this mode, ANSYS FLUENT will simply turn VTR recording on, play the animation, then stop the recording.

Frame-By-Frame
is used to produce a higher-quality video animation by recording one frame at a time. For each animation frame, this method will 1) play the frame on the screen (and generate the picture file, if needed), 2) preroll the VTR, and 3) record the frame. If the animation has 50 frames, this procedure is repeated 50 times, i.e., 50 record passes are made. This is the recommended method, because the real-time playback of the animation will usually be too slow and choppy.

Frames/Pass
is used in conjunction with **Frame-By-Frame** recording to try and speed up this process, if possible. It specifies the number of animation frames recorded to tape per record pass. If the animation is long enough (200 frames or more), you can try setting this value to 2 or higher. For example, if you set this value to 2 for a 202-frame animation, it will record animation frame 1 during the first pass, frames 2 and 102 during the second pass, frames 3 and 103 during the third pass, and so on. This is possible only if the animation frames can be rendered in time to be inserted onto the tape during a record pass, so use this setting with caution.

Frame Hold Counts
specifies the number of video frames to hold an animation frame during recording. The video standard NTSC has a frame rate of 30 frames/sec. At that rate, a 150-frame animation will take only 5 seconds to play. To stretch out the animation, you can record the same animation frame over 2 or more video frames.

Begin Hold
specifies the number of video frames to hold the first animation frame. It helps to hold the first frame for about 5 seconds (150 video frames) so that the viewer can get accustomed to the picture before the animation begins.

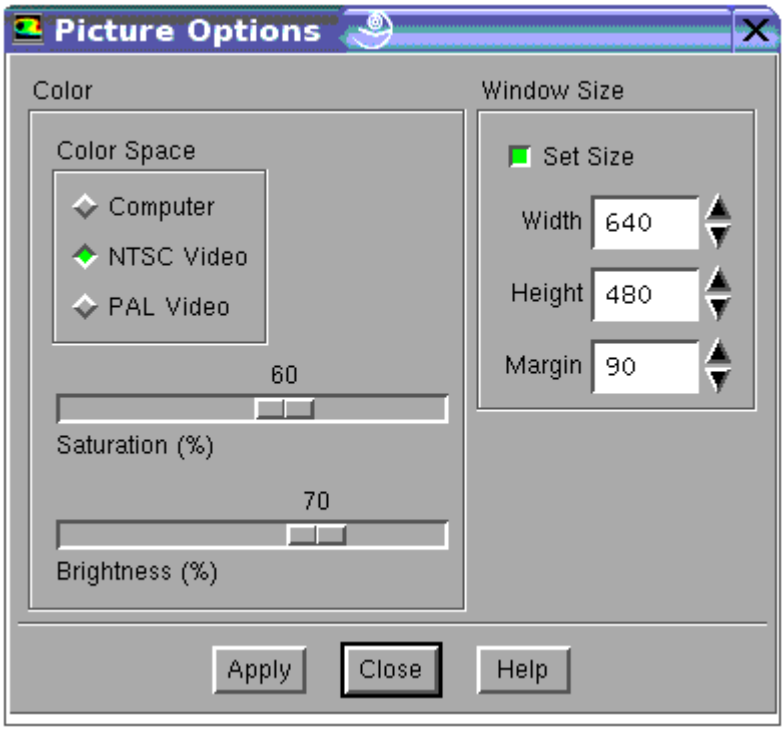
Frame Hold
specifies the number of video frames to hold each animation frame, other than the first and last. To slow down your recorded animation, try setting this value to 2 or 3.

End Hold
specifies the number of video frames to hold the last animation frame. You may want to hold the last animation frame for about 5 seconds to provide closure.

36.7.15.5. Picture Options Dialog Box

The **Picture Options** dialog box allows you to adjust the color levels and size of the picture in the graphics window. This dialog box has an **Apply** push button, so as you adjust levels, you can click on **Apply** to see the effect on the picture in the current graphics window. Each option has a presetting, so you can click on **Apply** immediately to see the effect of the presettings.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Color

contains options for adjusting the color levels of the picture.

Color Space

lets you choose a filter that will restrict all colors to within a particular color range by modifying individual color values, if needed.

Computer

when selected, effectively turns off color filtering.

NTSC Video

when selected, restricts colors to the NTSC color space. NTSC (National Television Standards Committee) is the main video format used in North America and Japan.

PAL Video

when selected, restricts colors to the PAL color space. PAL (Phase Alternate Line) is the video format used by some countries in Western Europe, Asia, and Australia.

Saturation (%)

controls the percentage of color saturation (or purity) of the colors in the picture. A saturation in the range of 0-20% produces grayed-out colors. A saturation in the range of 40-60% produces pastel colors. A saturation in the range of 80-100% produces vivid colors. For video, this value should be no more than 80%.

Brightness (%)

controls the color brightness of the picture. When lowering the saturation, it is often desirable to lower the brightness a little bit as well to avoid pastel colors.

Window Size

allows you to resize the graphics window so that you can match up the size to your VTR device.

Set Size

lets you decide whether or not you want the **Width**, **Height**, and **Margin** values to be used to resize the graphics window.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Width
specifies the width in pixels of the new window size.

Height
specifies the height in pixels of the new window size.

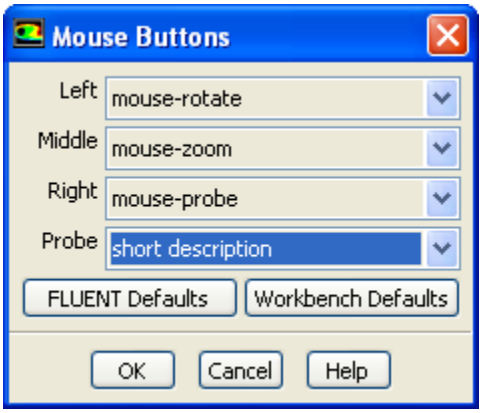
Margin
specifies a margin in pixels to create around the picture. This can help you produce a video image that does not include unwanted parts of the screen, such as the window border.

36.7.16. Display/Mouse Buttons...

The **Display/Mouse Buttons...** menu item opens the *Mouse Buttons Dialog Box* (p. 2157).

36.7.16.1. Mouse Buttons Dialog Box

The **Mouse Buttons** dialog box is used to set the actions taken when one of the mouse buttons is clicked in a graphics window. See *Controlling the Mouse Button Functions* (p. 1427) for details.



Controls

Left
sets the function associated with the left button. In 2D the default setting is **mouse-dolly**, and in 3D it is **mouse-rotate**.

Middle
sets the function associated with the middle button. The default setting for both 2D and 3D is **mouse-zoom**.

Right
sets the function associated with the right button. This button is not used on a two button mouse. The default setting for both 2D and 3D is **mouse-probe**.

Probe
turns the probe function on and off.

FLUENT Defaults
sets the left, middle, right, and probe buttons as described above.

Workbench Defaults
sets the mouse buttons as follows:

Left
In 2D and 3D the default setting is **mouse-dolly**.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Middle
In 2D the default setting is **mouse-rotate** and in 3D it is **mouse-probe**.

Right
In 2D and 3D the default setting is **mouse-zoom**.

Probe
turns the probe function on and off.

36.8. Report Menu

For additional information, please see the following sections:

36.8.1. Report/Result Reports...
36.8.2. Report/Input Summary...
36.8.3. Report/S2S Information...
36.8.4. Report/Reference Values...

36.8.1. Report/Result Reports...

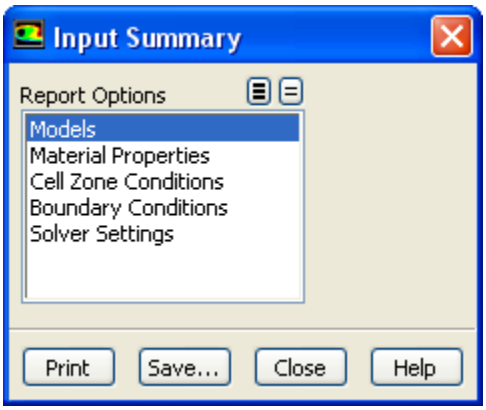
The **Report/Result Reports...** menu item opens the *Reports Task Page* (p. 2026).

36.8.2. Report/Input Summary...

The **Report/Input Summary...** menu item opens the *Input Summary Dialog Box* (p. 2158).

36.8.2.1. Input Summary Dialog Box

The **Input Summary** dialog box allows you to report the current settings for physical models, boundary conditions, material properties, and solution parameters. See *Generating a Summary Report* (p. 1525) for details about the items below.



Controls

Report Options
contains a selectable list of the information that is available for the report.

Print
prints the selected information to the console.

Save...
opens the *The **Select File** Dialog Box* (p. 33), in which you can specify the filename under which to save the output of the summary report.

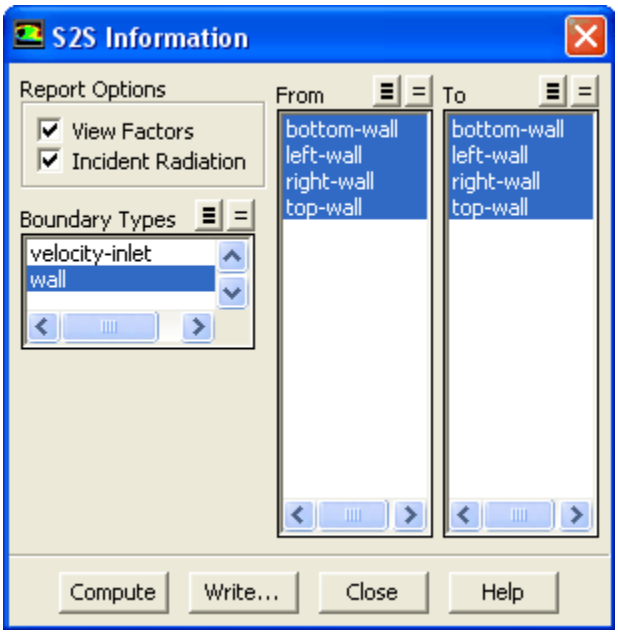
Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.8.3. Report/S2S Information...

The **Report/S2S Information...** menu item opens the *S2S Information Dialog Box* (p. 2159).

36.8.3.1. S2S Information Dialog Box

The **S2S Information** dialog box allows you to report values of the view factor and radiation emitted from one zone to any other zone. See *Reporting Radiation in the S2S Model* (p. 739) for details about the items below.



Controls

Report Options
contains items for which information is available for reporting.

View Factors
turns on the computation of view factors from one zone to the other.

Incident Radiation
turns on the computation of the incident radiation from one zone to the other.

Boundary Types
contains a selectable list of types of boundary zones. If you select (or deselect) an item in this list, all zones of that type will be selected (or deselected) automatically in the **From** and **To** lists.

From
contains a selectable list of boundary zones for which you would like data reported from the selected zone.

To
contains a selectable list of boundary zones for which you would like data reported to the selected zone.

Compute
computes the view factors and/or incident radiation on the selected zones.

Write...
opens the *The **Select File** Dialog Box* (p. 33), which you can use to save the data as an S2S Info File (`.sif` format).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.8.4. Report/Reference Values...

The **Report/Reference Values...** menu item opens the *Reference Values Task Page* (p. 1898).

36.9. Parallel Menu

For additional information, please see the following sections:

36.9.1. Parallel/Auto Partition...
36.9.2. Parallel/Partitioning and Load Balancing...
36.9.3. Parallel/Thread Control...
36.9.4. Parallel/Network/Database...
36.9.5. Parallel/Network/Configure...
36.9.6. Parallel/Network/Show Connectivity...
36.9.7. Parallel/Network/Show Latency
36.9.8. Parallel/Network/Show Bandwidth
36.9.9. Parallel/Timer/Usage
36.9.10. Parallel/Timer/Reset

36.9.1. Parallel/Auto Partition...

The **Parallel/Auto Partition...** menu item opens the *Auto Partition Mesh Dialog Box* (p. 2160).

36.9.1.1. Auto Partition Mesh Dialog Box

The **Auto Partition Mesh** dialog box allows you to set the parameters for automatic partitioning when reading an unpartitioned mesh into the parallel solver. See *Partitioning the Mesh Automatically* (p. 1604) for details.

Controls

Method

contains a drop-down list of the recursive partition methods that can be used to create the mesh partitions. The choices include the **Cartesian Axes**, **Cartesian Strip**, **Cartesian X-Coordinate**, **Cartesian Y-Coordinate**, **Cartesian Z-Coordinate**, **Cartesian R Axes**, **Cartesian RX-Coordinate**, **Cartesian RY-Coordinate**, **Cartesian RZ-Coordinate**, **Cylindrical Axes**, **Cylindrical R-Coordinate**, **Cylindrical Theta-Coordinate**, **Cylindrical Z-Coordinate**, **Metis**, **Polar Axes**, **Polar R-Coordinate**, **Polar Theta-Coordinate**, **Principal Axes**, **Principal Strip**, **Principal X-Coordinate**, **Principal Y-Coordinate**, **Principal Z-Coordinate**, **Spherical Axes**, **Spherical Rho-Coordinate**, **Spherical Theta-Coordinate**, and **Spherical Phi-Coordinate** techniques, which are described in *Mesh Partitioning Methods* (p. 1616).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Case File

allows you to use a valid existing partition section in a case file (i.e., one where the number of partitions in the case file divides evenly into the number of compute nodes). You need to turn off the **Case File** option only if you want to change other parameters in the **Auto Partition Mesh** dialog box.

Across Zones

allows partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly different amounts of computation during the solution phase, for example if the domain contains both solid and fluid zones.

Optimizations

contains a toggle button to activate pre-testing.

Pre-Test

instructs the solver to test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection. Note that this option is available only when you choose **Principal Axes** or **Cartesian Axes** as the partitioning method.

36.9.2. Parallel/Partitioning and Load Balancing...

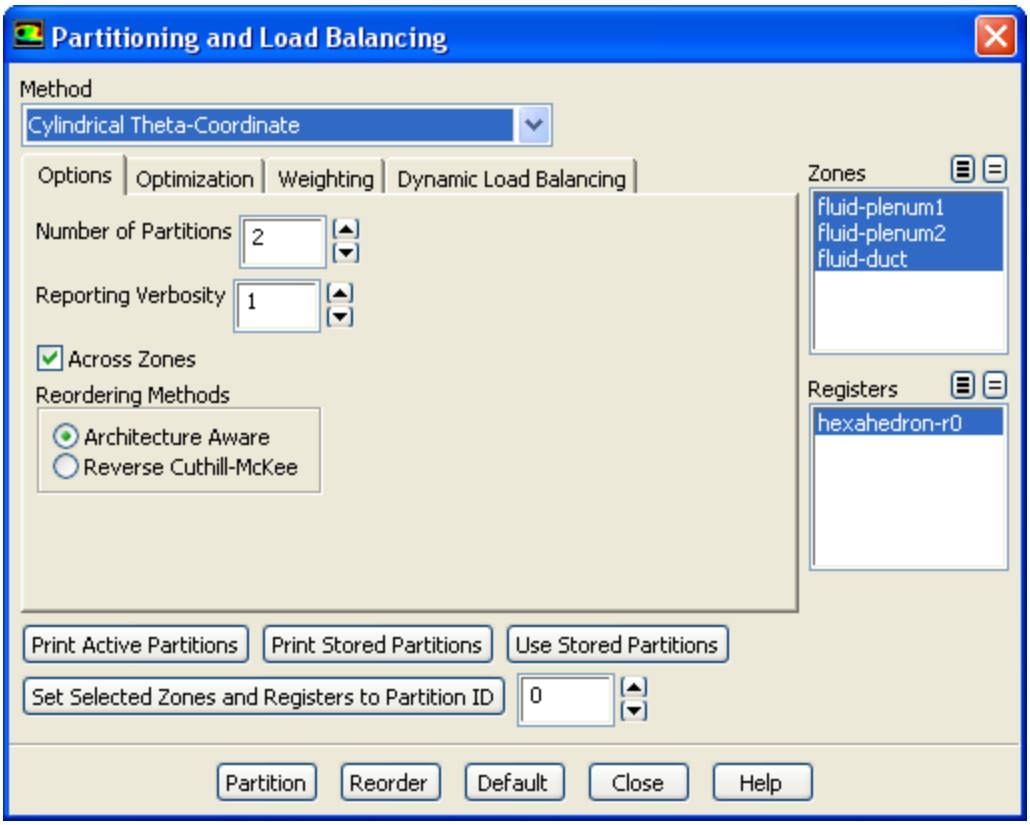
The **Parallel/Partitioning and Load Balancing...** menu item opens the *Partitioning and Load Balancing Dialog Box* (p. 2161).

36.9.2.1. Partitioning and Load Balancing Dialog Box

The **Partitioning and Load Balancing** dialog box allows you to partition the mesh into separate clusters of cells for separate processors on a parallel computer. Note that the **Partitioning and Load Balancing** dialog box is slightly different in the serial and parallel solvers. See *Partitioning the Mesh Manually and Balancing the Load* (p. 1605) for details.

The **Partitioning and Load Balancing** dialog box also allows you to enable and control ANSYS FLUENT's load balancing feature. Load balancing will automatically detect and analyze parallel performance, and re-distribute cells between the existing compute nodes to optimize it. See *Load Balancing* (p. 1614) for more information about load balancing.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Method

contains a drop-down list of the recursive partition methods that can be used to create the mesh partitions. The choices include the **Cartesian Axes**, **Cartesian Strip**, **Cartesian X-Coordinate**, **Cartesian Y-Coordinate**, **Cartesian Z-Coordinate**, **Cartesian R Axes**, **Cartesian RX-Coordinate**, **Cartesian RY-Coordinate**, **Cartesian RZ-Coordinate**, **Cylindrical Axes**, **Cylindrical R-Coordinate**, **Cylindrical Theta-Coordinate**, **Cylindrical Z-Coordinate**, **Metis**, **Polar Axes**, **Polar R-Coordinate**, **Polar Theta-Coordinate**, **Principal Axes**, **Principal Strip**, **Principal X-Coordinate**, **Principal Y-Coordinate**, **Principal Z-Coordinate**, **Spherical Axes**, **Spherical Rho-Coordinate**, **Spherical Theta-Coordinate**, and **Spherical Phi-Coordinate** techniques, which are described in *Mesh Partitioning Methods* (p. 1616).

Options

tab

Number of Partitions

defines the desired number of mesh partitions. This usually matches the number of processors available for parallel computing.

Reporting Verbosity

specifies the amount of information to be reported in the text (console) window during the partitioning. With the default value of 1, the solver will print the number of partitions created, the number of bisections performed, the time required for the partitioning, and the minimum and maximum cell, face, interface, and face-ratio variations. (See *Interpreting Partition Statistics* (p. 1622) for details.) If you increase the **Reporting Verbosity** to 2, the partition method used, the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each partition will also be printed in the console window. If you decrease the **Reporting Verbosity** to 0, only the number of partitions created and the time required for the partitioning will be reported.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Across Zones
allows partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly different amounts of computation during the solution phase, for example if the domain contains both solid and fluid zones.

Reordering Method
is used to optimize parallel performance

Architecture Aware
is the default option and it accounts for the system architecture and network topology in remapping the partitions to the processors.

Reverse Cuthill-McKee
minimizes the bandwidth of the compute-node connectivity matrix (the maximum distance between two connected processes) without incorporating the system architecture.

Optimization
tab contains toggle buttons for activating schemes to optimize the partitions created by the selected partition method. In addition, the optimization scheme will be applied until appropriate criteria are met, or the maximum number of iterations have been executed. If the **Iterations** counter is set to 0, the optimization scheme will be applied until completion, without limit on the maximum number of iterations.

Merge
attempts to decrease the number of interfaces by eliminating orphan cell clusters (an orphan cluster is a group of connected cells whose members each have at least one face coincident with an interface boundary).

Smooth
attempts to minimize the number of interfaces by sacrificing cells on the partition boundary to the neighboring partition to reduce the partition boundary surface area.

Pre-Test
instructs the solver to test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection. Note that this option is available only when you choose **Principal Axes** or **Cartesian Axes** as the partitioning method.

Weighting
tab allows you to set the appropriate weights, prior to partitioning the mesh.

Weight Types
allows you to turn on/off a specific weight. Weight types include: **Faces per Cell** (which is enabled by default, with additional weighting of **2**), **Solid Zones** (available only if solid cell zones are defined); **VOF** (available only if the volume of fluid multiphase model is turned on); **DPM** (available only for discrete phase simulations with injections defined); and **ISAT**.

User-Specified
allows you to use the default value (by not selecting the check box), or to specify a value yourself (by selecting the check box and entering a numerical value in the **Value** field.

Value
allows you to enter a user-specified numerical value that either defines the ratio used in the weighting, or (for the **Faces per Cell** weighting) is added to the calculated weight of each cell.

Dynamic Load Balancing
tab allows you to set load balancing thresholds and intervals.

Physical Models
allows you to set thresholds and intervals for the simulation's physical models.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Threshold
is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Interval
allows you to specify the desired interval for load balancing cycles, in terms of number of iterations. When a value of 0 is specified, ANSYS FLUENT will internally determine the best value to use, initially using an interval of 25 iterations. You can override this behavior by specifying a non-zero value. ANSYS FLUENT will then attempt to perform load balancing after every N iterations, where N is the specified **Balance Interval**.

Dynamic Mesh
allows you to set load balancing threshold and interval values for your dynamic mesh. This option is only available when the dynamic mesh model is enabled.

Auto
allows a percentage of interface faces and loads to be automatically traced.

Threshold
is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Interval
allows you to specify the desired interval for load balancing cycles, in terms of number of iterations. When a value of 0 is specified, ANSYS FLUENT will internally determine the best value to use, initially using an interval of 25 iterations. You can override this behavior by specifying a non-zero value. ANSYS FLUENT will then attempt to perform load balancing after every N iterations, where N is the specified **Balance Interval**.

Mesh Adaption
allows you to set threshold values for mesh adaption.

Threshold
is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Zones
contains a list of cell zones. Partitioning will be applied to cells in zones selected from this list.

Registers
contains a list of cell registers that have been created using the adaption tools. You can restrict partitioning to a group of cells by selecting a register containing the cells. See *Partitioning Within Zones or Registers* (p. 1613) for details.

Print Active Partitions
(parallel solver only) prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active partition (see *Checking the Partitions* (p. 1622) for information about active partitions) in the console window. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Print Stored Partitions
(parallel solver only) prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each stored partition (see *Checking the Partitions* (p. 1622) for information about stored partitions) in the console window. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Print Partitions

(serial solver only) prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each stored partition in the console window. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Set Selected Zones and Registers to Partition ID

allows you to set a value which assigns selected **Zones** and/or **Registers** to a specific partition ID. A region or zone is marked before setting the marked cells to one of the partition IDs.

Use Stored Partitions

(parallel solver only) allows you to make the stored cell partitions the active cell partitions. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

Partition

subdivides the mesh into the selected number of partitions using the prescribed method and optimization(s).

Default

sets all controls to their default values, as assigned by ANSYS FLUENT. After execution, the **Default** button becomes the **Reset** button.

Reset

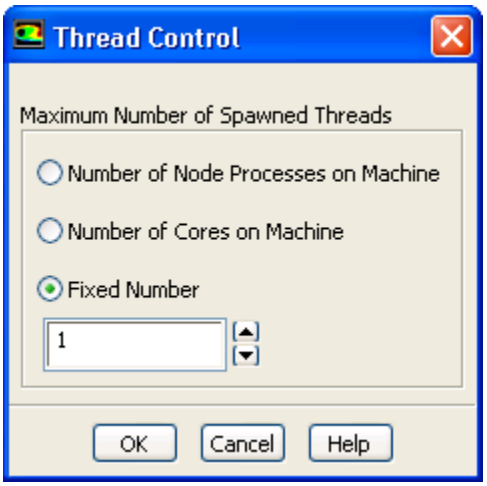
resets the fields to their most recently saved values (i.e., the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

36.9.3. Parallel/Thread Control...

The **Parallel/Thread Control...** menu item opens the *Thread Control Dialog Box* (p. 2165).

36.9.3.1. Thread Control Dialog Box

The **Thread Control** dialog box allows you to control the maximum number of threads on each machine, as described in *Controlling the Threads* (p. 1625).



Controls

Maximum Number of Spawned Threads

contains a list of options for defining the maximum number of threads on each machine.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Node Processes on Machine
specifies that the maximum number of threads on each machine is equal to the number of ANSYS FLUENT node processes on each machine.

Number of Cores on Machine
specifies that the maximum number of threads on each machine is equal to the number of cores on the machine. ANSYS FLUENT obtains the number of cores from the OS.

Fixed Number
specifies that the maximum number of threads that can be spawned on each machine is equal to the number you provide in the number-entry box below **Fixed Number**.

36.9.4. Parallel/Network/Database...

The **Parallel/Network/Database...** menu item opens the *Hosts Database Dialog Box* (p. 2166).

Important

The ability to read a hosts file using the **Hosts Database** dialog box is only available when using the `net` option (e.g., `-mpi=net`). While this feature is still available in ANSYS FLUENT 6.3, its use is discouraged.

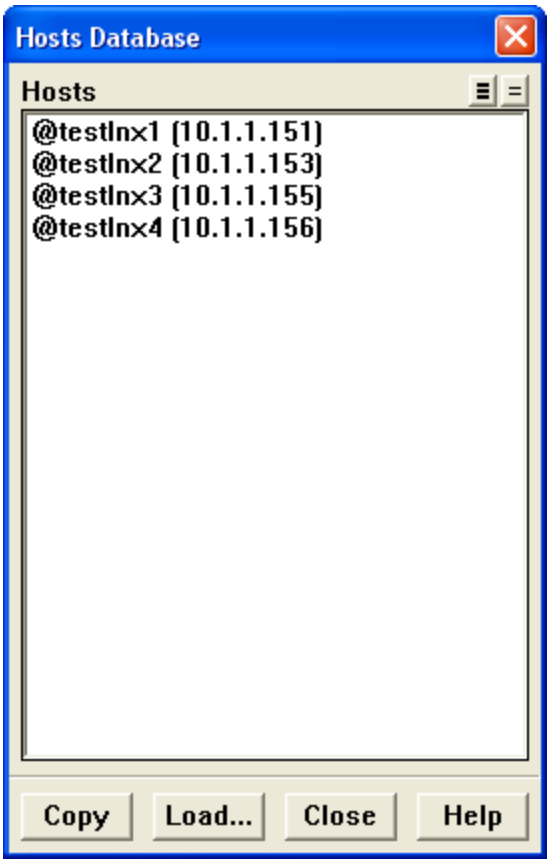
36.9.4.1. Hosts Database Dialog Box

When you are creating a parallel network of workstations, it is convenient to start with a list of machines that are part of your local network (a "hosts file"). You can load a file containing these names into the hosts database and then select the hosts that are available for creating a parallel configuration (or network) on a cluster of workstations.

The **Hosts Database** dialog box allows you to read a hosts file and copy selected entries to the list of available hosts in the *Network Configuration Dialog Box* (p. 2168).

Parallel → Network → **Database...**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Hosts

contains a selectable list of local host names. The default list consists of the hosts in your `fluent.hosts` or `.fluent.hosts` file, if you have one in your home directory. You can load other hosts files using the **Load** button.

If the hosts file exists in your home directory, its contents are automatically added to the hosts database at startup. Otherwise, the hosts database will be empty until you read in a hosts file.

Once the contents of the hosts file have been read, the host names will appear in the **Hosts** list. (ANSYS FLUENT will automatically add the IP (Internet Protocol) address for each recognized machine. If a machine is not currently on the local network, it will be labeled **unknown**.)

Copy

copies selected entries to the list of available hosts in the *Network Configuration Dialog Box* (p. 2168) on which you can spawn nodes.

Load...

opens the *The **Select File** Dialog Box* (p. 33), which allows you to read a hosts file containing a list of local host names.

36.9.5. Parallel/Network/Configure...

The **Parallel/Network/Configure...** menu item opens the *Network Configuration Dialog Box* (p. 2168).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

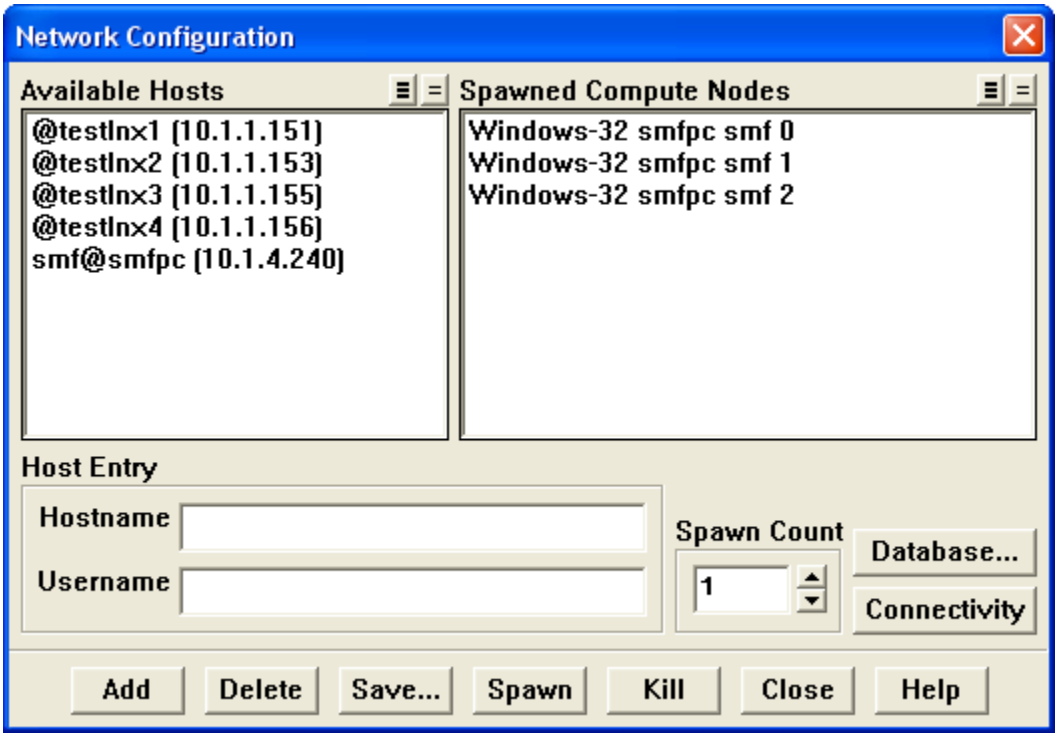
The ability to manually spawn additional compute nodes before reading the case file using the **Network Configuration** dialog box is only available when using the `net` option (e.g., `-mpi=net`). While this feature is still available in ANSYS FLUENT 6.3, its use is discouraged.

36.9.5.1. Network Configuration Dialog Box

Compute nodes are labeled sequentially starting at 0. In addition to the compute node processes, there is one host process. The host process is automatically started when ANSYS FLUENT starts, and it is killed when ANSYS FLUENT exits. It cannot be killed while running. Compute nodes, however, can be killed at any time, with the exception that compute node 0 can only be killed if it is the last remaining compute node process. The host process always spawns compute node 0. Compute node 0 spawns all other compute nodes.

If you want to spawn compute nodes on several different machines, or if you want to make any changes to the current network configuration (e.g., if you accidentally spawned too many compute nodes on the host machine when you started the solver), the *Network Configuration Dialog Box* (p. 2168) allows you to control the configuration of your parallel network.

Parallel → Network → **Configure...**



Controls

Available Hosts

contains a selectable list of hosts available for creating a parallel machine. If the hosts file `fluent.hosts` exists in your home directory, its contents are automatically added to the **Available Hosts** list at startup. Hosts can be added to the list either by specifying a hostname and optional username in the **Host Entry** box, or by copying selected hosts from the *Hosts Database Dialog Box* (p. 2166).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spawned Compute Nodes
contains the list of all compute node processes that form the parallel machine. Each entry lists the operating system, hostname, username, and compute node ID, in that order. Compute nodes can be added to this list by selecting hosts from the **Available Hosts** list.

Host Entry
is used to manually add and delete hosts from the **Available Hosts** list.

Hostname
is the internet name of a remote machine.

Username
is your login name on the machine specified in the **Hostname** field. If all your accounts have the same login name, you do not need to specify a username.

Spawn Count
defines the number of compute node processes to spawn on each selected host in the **Available Hosts** list.

Database...
opens the *Hosts Database Dialog Box* (p. 2166).

Connectivity
displays the network connectivity of all compute nodes selected in the **Spawned Compute Nodes** list. If no compute nodes are selected, the *Parallel Connectivity Dialog Box* (p. 2171) is opened.

Add
adds a workstation from the **Host Entry** box to the **Available Hosts** list.

To add a host to the **Available Hosts** list manually, you can enter the internet name of the remote machine in the **Hostname** field under **Host Entry**, enter your login name on that machine in the **Username** field (unless your accounts all have the same login name, in which case you need not specify a username), and then click the **Add** button. The specified host will be added to the **Available Hosts** list.

Delete
removes the host specified in the **Host Entry** box or selected in the **Available Hosts** list from the **Available Hosts** list.

To delete a host from the **Available Hosts** list in the *Network Configuration Dialog Box* (p. 2168), select the host and click the **Delete** button. The host name will be removed from the **Available Hosts** list (but the *Hosts Database Dialog Box* (p. 2166) will not be affected).

Save...
opens *The **Select File** Dialog Box* (p. 33), which allows you to write a hosts file for future use that contains all entries in the **Available Hosts** list. In a future session, you can load the contents of this file into the *Hosts Database Dialog Box* (p. 2166) and then copy the hosts over to the *Network Configuration Dialog Box* (p. 2168) in order to reproduce the current **Available Hosts** list.

Spawn
creates compute node processes on all hosts selected in the **Available Hosts** list.

Kill
kills the compute node processes selected in the **Spawned Compute Nodes** list.

Important

Remember that compute node 0 can only be killed if it is the last remaining compute node process.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The basic steps for spawning compute nodes are as follows:

1. Choose the host machine(s) on which to spawn compute nodes in the **Available Hosts** list. If the desired machine is not listed, you can use the **Host Entry** fields to manually add a host, or you can copy the desired host from the host database.
2. Set the number of compute node processes to spawn on each selected host machine in the **Spawn Count** field.
3. Click the **Spawn** button and the new node(s) will be spawned and added to the **Spawned Compute Nodes** list.

Common Problems Encountered During Node Spawning

The spawning process will try to establish a connection with a new compute node, but if after 50 seconds it receives no response from the new compute node, it will assume the spawn was unsuccessful. The spawn will be unsuccessful, for example, if the remote machine is unable to find the ANSYS FLUENT executable. To manually test if the spawning machine can start a new compute node, you can type

`rsh [-l`*username*`]` *hostname* `fluent -t0 -v`

from a shell prompt on the spawning machine. *hostname* should be replaced with the internet name of the machine on which you want to spawn a compute node, and *username* should be replaced with your login name on the remote machine specified by *hostname*.

Important

If all your accounts have the same login name, you do not need to specify a username. (The square brackets around `-l`*username* indicate that it is not always required; if you do enter a login name, do not include the square brackets.) Note that on some systems, the remote shell command is `remsh` instead of `rsh`.

The spawn test could fail for several reasons:

Login incorrect.
The machine spawning a new compute node must be able to `rsh` to the machine where the new process will reside, or the spawn will fail. There are several ways to enable this capability. Consult your systems administrator for assistance.

fluent: Command not found.
The `rsh` to the remote machine succeeded, but the path to the ANSYS FLUENT shell script could not be found on that machine. If you are using `csh`, then the path to the ANSYS FLUENT shell script should be added to the `path` variable in your `.cshrc` file. If that also fails, you can use the `parallel/network/path` text command to set the path to the `Fluent.Inc` installation directory directly before spawning the compute node.

`parallel` → `network` → `path`

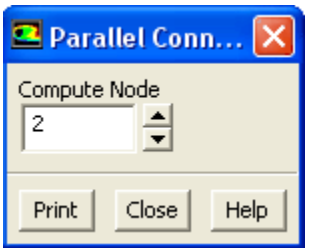
36.9.6. Parallel/Network/Show Connectivity...

The **Parallel/Network/Show Connectivity...** menu item opens the *Parallel Connectivity Dialog Box* (p. 2171).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.9.6.1. Parallel Connectivity Dialog Box

The **Parallel Connectivity** dialog box prints the connectivity of the selected compute node. See *Checking Network Connectivity* (p. 1625) for details.



Controls

Compute Node
indicates the compute node ID for which connectivity information is desired.

Print
prints information (in the console window) about the network connectivity for the selected compute node.

36.9.7. Parallel/Network/Show Latency

The **Parallel/Network/Show Latency** menu item prints information to the console about the communication speed for each node, as well as minimum and maximum latency between two nodes. See *Checking Latency and Bandwidth* (p. 1628) for details.

36.9.8. Parallel/Network/Show Bandwidth

The **Parallel/Network/Show Bandwidth** menu item prints information to the console about the amount of data communicated within one second between two nodes, as well as minimum and maximum bandwidth between two nodes. See *Checking Latency and Bandwidth* (p. 1628) for details.

36.9.9. Parallel/Timer/Usage

The **Parallel/Timer/Usage** menu item prints performance statistics in the console.

36.9.10. Parallel/Timer/Reset

The **Parallel/Timer/Reset** menu item clears the performance meter.

36.10. View Menu

For additional information, please see the following sections:

36.10.1. View/Toolbars
36.10.2. View/Navigation Pane
36.10.3. View/Task Page
36.10.4. View/Graphics Window
36.10.5. View/Embed Graphics Window
36.10.6. View/Show All
36.10.7. View/Show Only Console
36.10.8. View/Graphics Window Layout

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.10.9. View/Save Layout

36.10.1. View/Toolbars

The **View/Toolbars** menu item allows you to toggle the visibility of the toolbars in the application window.

36.10.2. View/Navigation Pane

The **View/Navigation Pane** menu item allows you to toggle the visibility of the navigation pane in the application window.

36.10.3. View/Task Page

The **View/Task Page** menu item allows you to toggle the visibility of the task pages in the application window.

36.10.4. View/Graphics Window

The **View/Graphics Window** menu item allows you to toggle the visibility of the graphics window in the application window. This option is only visible when the graphics window is embedded in the application.

36.10.5. View/Embed Graphics Window

The **View/Embed Graphics Window** menu item allows you to toggle between attaching and detaching the graphics window within the application.

36.10.6. View/Show All

The **View/Show All** menu item allows you to display the toolbars, the navigation pane, the task pages, and the graphics window, if one or more of them is currently not visible.

36.10.7. View/Show Only Console

The **View/Show Only Console** menu item allows you to display only the console window in the application.

36.10.8. View/Graphics Window Layout

The **View/Graphics Window Layout** menu item displays several layout options for use with multiple graphics windows. This option is only visible when the graphics window is embedded in the application.

36.10.9. View/Save Layout

The **View/Save Layout** menu item is used to save the current arrangement of dialog boxes and graphics windows. The positions of these items on your screen will be written to a `.cxlayout` file in your home folder.

36.11. Turbo Menu

For additional information, please see the following sections:

36.11.1. Turbo/Report...
36.11.2. Turbo/Averaged Contours...
36.11.3. Turbo/2D Contours...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

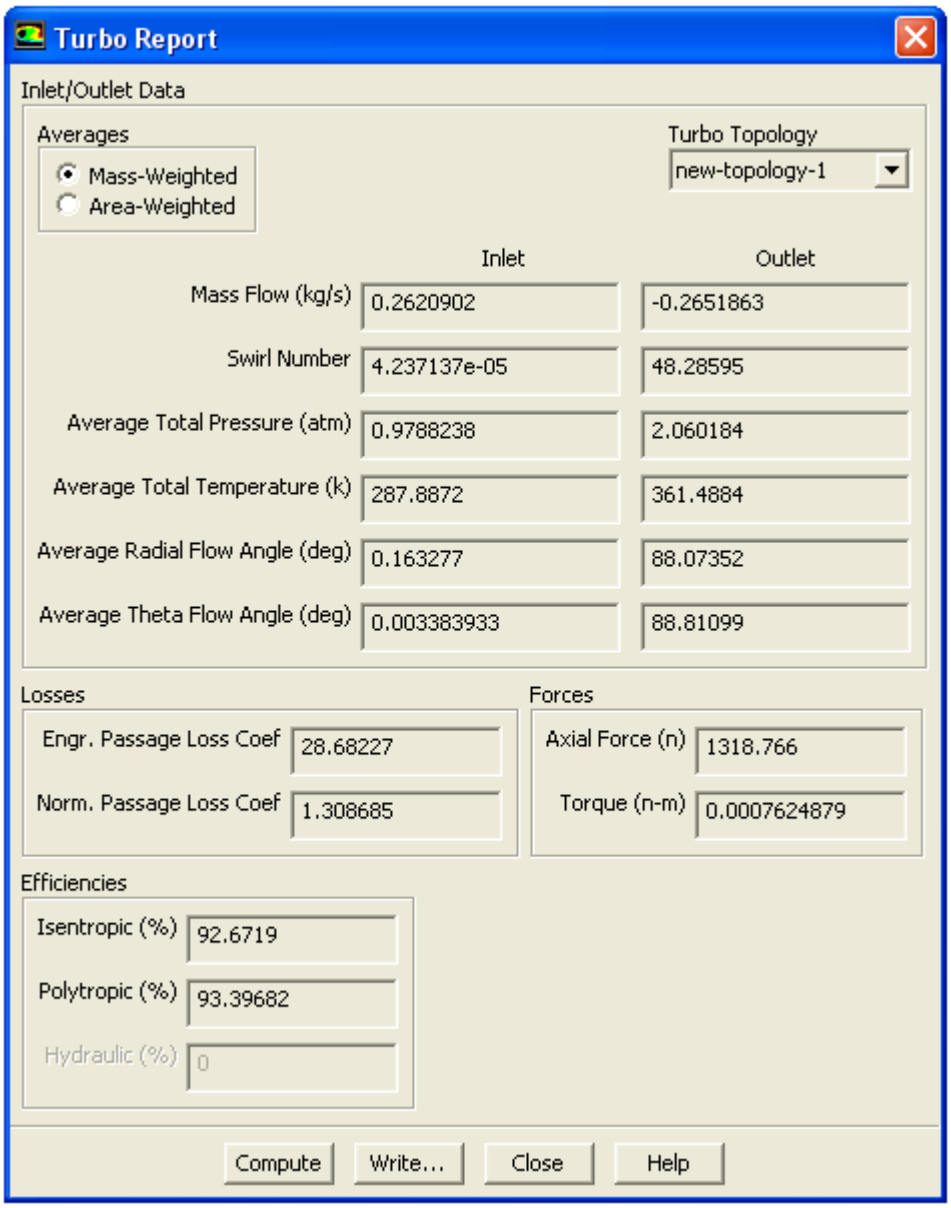
36.11.4. Turbo/Averaged XY Plot...
36.11.5. Turbo/Options...

36.11.1. Turbo/Report...

The **Turbo/Report...** menu item opens the *Turbo Report Dialog Box* (p. 2173).

36.11.1.1. Turbo Report Dialog Box

The **Turbo Report** dialog box allows you to calculate turbomachinery-specific quantities and integrals. See *Generating Reports of Turbomachinery Data* (p. 1487) for details.



Controls

Inlet/Outlet Data

contains quantities that can be calculated at inlets and outlets.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Averages
allows you to choose between **Mass-Weighted** and **Area-Weighted** averages for all applicable computed quantities. These quantities are calculated for the **Inlet** and **Outlet** topologies where applicable.

Turbo Topology
contains a list of defined topologies. Select from the list to display the values for the selected topology.

Mass Flow
is the mass flow rate through a surface as defined in *Equation 31–2* (p. 1488).

Swirl Number
is the swirl number as defined in *Equation 31–3* (p. 1488).

Average Total Pressure
is the area-averaged or mass-averaged total pressure as defined in *Equation 31–5* (p. 1489) or *Equation 31–6* (p. 1489).

Average Total Temperature
is the area-averaged or mass-averaged total temperature as defined in *Equation 31–7* (p. 1489) or *Equation 31–8* (p. 1489).

Average Radial Flow Angle
is the area-averaged or mass-averaged radial flow angle as defined in *Equation 31–9* (p. 1490) or *Equation 31–11* (p. 1490).

Average Theta Flow Angle
is the area-averaged or mass-averaged tangential flow angle as defined in *Equation 31–10* (p. 1490) or *Equation 31–12* (p. 1490).

Losses
contains the values of loss-related coefficients.

Engr. Passage Loss Coef
is the engineering loss coefficient as defined in *Equation 31–13* (p. 1490).

Norm. Passage Loss Coef
is the normalized loss coefficient as defined in *Equation 31–14* (p. 1491).

Forces
contains the axial force and the torque on the rotating parts.

Axial Force
is the axial force on the rotating parts as defined in *Equation 31–15* (p. 1491).

Torque
is the torque on the rotating parts as defined in *Equation 31–16* (p. 1491).

Efficiencies
contains the values of isentropic, polytropic and hydraulic efficiencies.

Isentropic
is the isentropic efficiency for a compressor or a turbine (motor) calculated in the presence of a compressible working fluid as defined in *Equation 31–21* (p. 1493) or *Equation 31–27* (p. 1495).

Polytropic
is the polytropic efficiency for a compressor or a turbine (motor) calculated in the presence of a compressible working fluid as defined in *Equation 31–22* (p. 1494) or *Equation 31–28* (p. 1496).

Hydraulic
is the hydraulic efficiency for a pump or a hydraulic turbine (motor) calculated in the presence of an incompressible working fluid as defined in *Equation 31–17* (p. 1492) or *Equation 31–23* (p. 1494).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute
starts the calculation of the quantities in all the fields in the **Turbo Report** dialog box. Note that this process may take some time for a large problem.

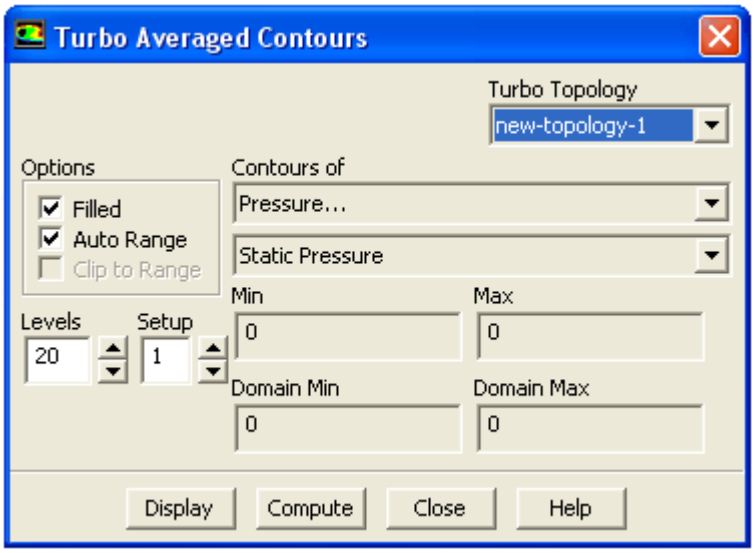
Write...
opens the **Select File** dialog box, which you can use to save the reported values to a file.

36.11.2. Turbo/Averaged Contours...

The **Turbo/Averaged Contours...** menu item opens the *Turbo Averaged Contours Dialog Box* (p. 2175).

36.11.2.1. Turbo Averaged Contours Dialog Box

The **Turbo Averaged Contours** dialog box allows you display turbomachinery-specific circumferentially averaged contours of variables projected on an r- z plane. See *Displaying Turbomachinery Averaged Contours* (p. 1496) for details.



Controls

Turbo Topology
contains a list of defined topologies. Select from the list to display the values for the selected topology.

Options
contains the check buttons that set various contour display options.

Filled
toggles between filled contours and line contours.

Auto Range
toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range
determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See *Specifying the Range of Magnitudes Displayed* (p. 1390) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Levels
sets the number of contour levels that are displayed.

Setup
indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the dialog box. See *Storing Contour Plot Settings* (p. 1393) for details.

Contours of
contains a list from which you can select the scalar field to be contoured.

Min
shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Domain Min
shows the global minimum value of the scalar field for the entire domain.

Domain Max
shows the global maximum value of the scalar field for the entire domain.

Display
draws the contours in the active graphics window.

Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

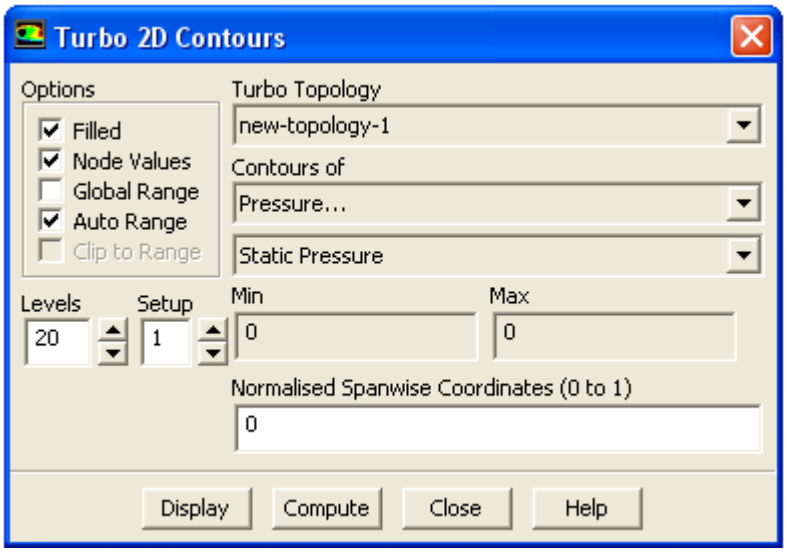
36.11.3. Turbo/2D Contours...

The **Turbo/2D Contours...** menu item opens the *Turbo 2D Contours Dialog Box* (p. 2176).

36.11.3.1. Turbo 2D Contours Dialog Box

The **Turbo 2D Contours** dialog box allows you display turbomachinery-specific contours of variables on surfaces of constant pitchwise, spanwise, or meridional coordinates, projected onto a plane. See *Displaying Turbomachinery 2D Contours* (p. 1497) for details.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Turbo Topology

contains a list of defined topologies. Select from the list to display the values for the selected topology.

Options

contains the check buttons that set various contour display options.

Filled

toggles between filled contours and line contours.

Node Values

toggles between using scalar field values at nodes and at cell centers for computing the contours.

Global Range

toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range

toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range

determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See *Specifying the Range of Magnitudes Displayed* (p. 1390) for details.

Levels

sets the number of contour levels that are displayed.

Setup

indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the dialog box. See *Storing Contour Plot Settings* (p. 1393) for details.

Contours of

contains a list from which you can select the scalar field to be contoured.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min
shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Normalised Spanwise Coordinates (0 to 1)
allows you to specify the coordinate for the spanwise surface you want to create.

Display
draws the contours in the active graphics window.

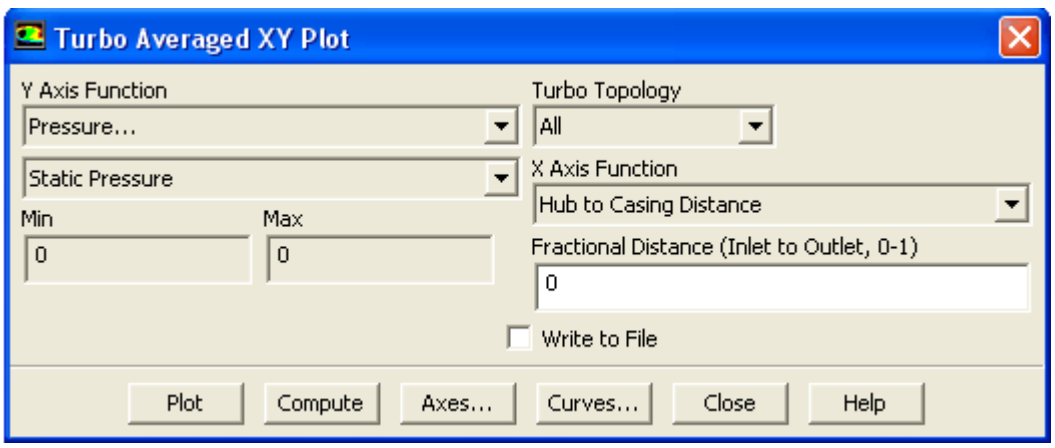
Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

36.11.4. Turbo/Averaged XY Plot...

The **Turbo/Averaged XY Plot...** menu item opens the *Turbo Averaged XY Plot Dialog Box* (p. 2178).

36.11.4.1. Turbo Averaged XY Plot Dialog Box

The **Turbo Averaged XY Plot** dialog box allows you to display data in an XY plot format as a function of either the meridional or the spanwise coordinate. See *Generating Averaged XY Plots of Turbomachinery Solution Data* (p. 1499) for details.



Controls

Y Axis Function
contains a list of solution variables that can be used for the y axis of the plot.

Min
shows the minimum value of the scalar field.

Max
shows the maximum value of the scalar field.

Turbo Topology
contains a list of defined topologies. Select from the list to display the values for the selected topology.

X Axis Function
allows you to select the coordinate to be used for the x axis of the plot. The choices are **Hub to Casing Distance** (spanwise coordinate), and **Meridional Distance** (meridional coordinate).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fractional Distance
sets a fractional value (0 to 1) for either the spanwise **Hub to Casing** distance or the meridional **Inlet to Outlet** distance, depending on your selection for **X Axis Function**.

Write to File
activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...**button will open the *The **Select File** Dialog Box* (p. 33), in which you can specify a name and save a file containing the plot data. The format of this file is described in *XY Plot File Format* (p. 1477).

Plot
plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes. If the **Write to File** option is turned on, this button becomes the **Write...** button.

Write...
opens the *The **Select File** Dialog Box* (p. 33), in which you can save the plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Compute
calculates the scalar field and updates the **Min** and **Max** values.

Axes...
opens the *Axes Dialog Box* (p. 2022), which allows you to customize the plot axes.

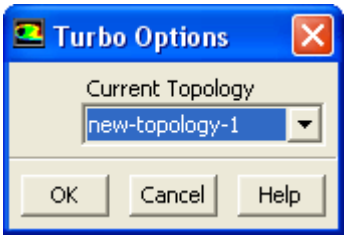
Curves...
opens the *Curves Dialog Box* (p. 2024), which allows you to customize the curves used in the XY plot.

36.11.5. Turbo/Options...

The **Turbo/Options...** menu item opens the *Turbo Options Dialog Box* (p. 2179).

36.11.5.1. Turbo Options Dialog Box

The **Turbo Options** dialog box allows you to globally set the turbomachinery topology for your model. See *Globally Setting the Turbomachinery Topology* (p. 1500) for details.



Controls

Current Topology
contains a list of predefined turbo topologies.

36.12. Help Menu

For additional information, please see the following sections:

36.12.1. Help/User's Guide Contents...
36.12.2. Help/User's Guide Index...
36.12.3. Help/PDF...
36.12.4. Help/Context-Sensitive Help
36.12.5. Help/Using Help...

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.12.6. Help/Online Technical Resources...
36.12.7. Help/License Usage
36.12.8. Help/Version...

36.12.1. Help/User's Guide Contents...

The **Help/User's Guide Contents...** menu item opens the help viewer to the contents page of the User's Guide. See *Using the GUI Help System* (p. 41) for details.

36.12.2. Help/User's Guide Index...

The **Help/User's Guide Index...** menu item opens the help viewer to the index page of the User's Guide. See *Using the GUI Help System* (p. 41) for details.

36.12.3. Help/PDF...

The **Help/PDF...** menu item displays a submenu that allows you access to the User's Guide and other manuals in printable format (PDF).

36.12.4. Help/Context-Sensitive Help

The **Help/Context-Sensitive Help** menu item allows you to get help for a specific topic represented by a pull-down menu item, a dialog box, or another part of the graphical user interface. When you select **Context-Sensitive Help**, the screen cursor will change to the shape of a question mark. Now you can select an item from a pull-down menu or click on another component of the graphical user interface. The web browser will open directly to the corresponding section of the User's Guide.

Important

Context-sensitive help is not available on Windows systems.

36.12.5. Help/Using Help...

The **Help/Using Help...** menu item opens the help viewer directly to the section on using the on-line help facility. See *Using the GUI Help System* (p. 41) for details.

36.12.6. Help/Online Technical Resources...

The **Help/Online Technical Resources...** menu item opens the web browser to the ANSYS Customer Portal web site where you can search for solutions and log requests. See *Using the GUI Help System* (p. 41) for details.

36.12.7. Help/License Usage

The **Help/License Usage...** menu item provide ANSYS FLUENT license information. See *Using the GUI Help System* (p. 41) for details.

36.12.8. Help/Version...

The **Help/Version...** menu item shows you the version and release of ANSYS FLUENT that you are using.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beta Features

ANSYS FLUENT contains Beta features that should be used under the supervision of an ANSYS support representative. Information about Beta features can be found on the ANSYS Customer Portal and the ANSYS FLUENT User Services Center.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix A. ANSYS FLUENT Model Compatibility

The following tables summarize the compatibility of several ANSYS FLUENT model categories:

- Multiphase Models (see *Modeling Multiphase Flows* (p. 1091))
- Moving Domain Models (See *Modeling Flows with Moving Reference Frames* (p. 513))
- Turbulence Models (See *Modeling Turbulence* (p. 643))
- Combustion Models (See Chapters *Modeling Species Transport and Finite-Rate Chemistry* (p. 799) – *Modeling Engine Ignition* (p. 929))

Note that a y indicates that two models are compatible with each other, while an n indicates that two models are not compatible with each other.

Table A.1 Multiphase Models vs. Moving Domain Models

	Sliding Mesh	Mixing Plane	Dynamic Mesh	Multiple Reference Frame	Single Reference Frame
Eulerian	y	y	y	y	y
VOF	y	y	y	y	y
Mixture	y	y	y	y	y
Discrete Phase	y	n	y	y	y

Table A.2 Multiphase Models vs. Turbulence Models

	Spalart–Allmaras	k–epsilon	k–omega	Reynolds Stress	LES
Eulerian	n	y	y	y	n
VOF	y	y	y	y	y
Mixture	y	y	y	y	n
Discrete Phase	y	y	y	y	y

Table A.3 Multiphase Models vs. Combustion Models

	Laminar Finite Rate	Eddy Dissipation	Eddy Dissipation Concept	Non–Premixed	Premixed	Partially Premixed	Composition PDF Transport	Pollutants
Eulerian	y	y	n	n	n	n	n	n
VOF	y	y	n	n	n	n	n	n
Mixture	y	y	n	n	n	n	n	n

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase	y	y	y	y	y	y	y	y

Table A.4 Moving Domain Models vs. Turbulence Models

	Spalart–Allmaras	k–epsilon	k–omega	Reynolds Stress	LES
Sliding Mesh	y	y	y	y	y
Mixing Plane	y	y	y	y	n
Dynamic Mesh	y	y	y	y	y
Multiple Reference Frame	y	y	y	y	y
Single Reference Frame	y	y	y	y	y

Table A.5 Moving Domain Models vs. Combustion Models

	Laminar Finite Rate	Eddy Dissipation	Eddy Dissipation Concept	Non–Premixed	Premixed	Partially Premixed	Composition PDF Transport	Pollutants
Sliding Mesh	y	y	y	y	y	y	y	y
Mixing Plane	n	n	n	n	n	n	n	n
Dynamic Mesh	y	y	y	y	y	y	y	y
Multiple Reference Frame	y	y	y	y	y	y	y	y
Single Reference Frame	y	y	y	y	y	y	y	y

Table A.6 Turbulence Models vs. Combustion Models

	Laminar Finite Rate	Eddy Dissipation	Eddy Dissipation Concept	Non–Premixed	Premixed	Partially Premixed	Composition PDF Transport	Pollutants
Spalart–Allmaras	y	n	n	n	n	n	n	n
k-epsilon	y	y	y	y	y	y	y	y
k–omega	y	y	y	y	y	y	y	y

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reynolds Stress	y	y	y	y	y	y	y	y
LES	y	y	y	y	y	y	y	y

Key:

n = not compatible

y = compatible

*** Includes Standard, RNG, and Realizable k–epsilon models**

**** Includes Standard and SST k–omega models**

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix B. Case and Data File Formats

This Appendix describes the contents and formats of ANSYS FLUENT case and data files.

- *Guidelines* (p. 2187)
- *Formatting Conventions in Binary and Formatted Files* (p. 2187)
- *Grid Sections* (p. 2188)
- *Other (Non-Grid) Case Sections* (p. 2199)
- *Data Sections* (p. 2201)

Descriptions of the sections are grouped according to function.

- *Formatting Conventions in Binary and Formatted Files* (p. 2187) : The differences between formatted and binary files.
- *Grid Sections* (p. 2188) : Creating grids for ANSYS FLUENT.
- *Other (Non-Grid) Case Sections* (p. 2199)
- *Grid Sections* (p. 2188) *Data Sections* (p. 2201) : Importing solutions into another postprocessor.

The case and data files may contain other sections that are intended for internal use only.

B.1. Guidelines

The ANSYS FLUENT case and data files are broken into several sections according to the following guidelines:

- Each section is enclosed in parentheses and begins with a decimal integer indicating its type. This integer is different for formatted and binary files (*Formatting Conventions in Binary and Formatted Files* (p. 2187)).
- All groups of items are enclosed in parentheses. This makes skipping to ends of (sub)sections and parsing them very easy. It also allows for easy and compatible addition of new items in future releases.
- Header information for lists of items is enclosed in separate sets of parentheses preceding the items, and the items are enclosed in their own parentheses.

B.2. Formatting Conventions in Binary and Formatted Files

For formatted files, examples of file sections are given in *Grid Sections* (p. 2188) and *Other (Non-Grid) Case Sections* (p. 2199). For binary files, the header indices described in this section (e.g., `10` for the node section) are preceded by `20` for single-precision binary data, or by `30` for double-precision binary data (e.g., `2010` or `3010` instead of `10`). The end of the binary data is indicated by `End of Binary Section 2010` or `End of Binary Section 3010` before the closing parameters of the section.

An example with the binary data represented by periods is as follows:

```
(2010 (2 1 2aad 2 3)(
 .
 .
 .
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
)
End of Binary Section 2010)
```

B.3. Grid Sections

Grid sections are stored in the case file. A grid file is a subset of a case file, containing only those sections pertaining to the grid. The currently defined grid sections are:

- Comment (See *Comment* (p. 2188))
- Header (See *Header* (p. 2188))
- Dimensions (See *Dimensions* (p. 2189))
- Nodes (See *Nodes* (p. 2189))
- Periodic Shadow Faces (See *Periodic Shadow Faces* (p. 2190))
- Cells (See *Cells* (p. 2191))
- Faces (See *Faces* (p. 2192))
- Face Tree (See *Face Tree* (p. 2194))
- Cell Tree (See *Cell Tree* (p. 2194))
- Interface Face Parents (See *Interface Face Parents* (p. 2195))

The section ID numbers are indicated in both symbolic and numeric forms. The symbolic representations are available as symbols in a Scheme source file (`xfile.scm`), which is available from ANSYS Inc., or as macros in a C header file (`xfile.h`), which is located in your installation area.

B.3.1. Comment

Index:	0
Scheme symbol:	`xf-comment`
C macro:	`XF_COMMENT`
Status:	optional

Comment sections can appear anywhere in the file (except within other sections) as:

```
(0 "comment text")
```

It is recommended to precede each long section, or group of related sections, by a comment section explaining what is to follow.

Example:

```
(0 "Variables:")
 (37 (
 (relax-mass-flow 1)
 (default-coefficient ())
 (default-method 0)
 ))
```

B.3.2. Header

Index:	1
Scheme symbol:	`xf-header`
C macro:	`XF_HEADER`

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Status: optional

Header sections can appear anywhere in the file (except within other sections). The following is an example:

```
(1 "TGrid 2.1.1")
```

The purpose of this section is to identify the program that wrote the file. Although it can appear anywhere, it is one of the first sections in the file. Additional header sections indicate other programs that may have been used in generating the file. It provides a history mechanism showing where the file came from and how it was processed.

B.3.3. Dimensions

Index: 2
Scheme symbol: `xf-dimension`
C macro: `XF_DIMENSION`
Status: optional

The dimensions of the grid appear as:

```
(2 ND)
```

where ND is 2 or 3. This section is supported as a check that the grid has the appropriate dimension.

B.3.4. Nodes

Index: 10
Scheme symbol: `xf-node`
C macro: `XF_NODE`
Status: required

Format:

```
(10 (zone-id first-index last-index type ND)(
 x1 y1 z1
 x2 y2 z2
 .
 .
 .
 ))
```

- If `zone-id` is zero, this provides the total number of nodes in the grid. `first-index` will then be one, `last-index` will be the total number of nodes *in hexadecimal*, `type` is equal to `1`, NDis the dimensionality of the grid, and there are no coordinates following (the parentheses for the coordinates are omitted as well).

  For example: `(10 (0 1 2d5 1 2))`

- If `zone-id` is greater than zero, it indicates the zone to which the nodes belong. `first-index` and `last-index` are the indices of the nodes in the zone, *in hexadecimal*. The values of `last-index` in each zone must be less than or equal to the value in the declaration section. **Type** is always equal to `1`.

  NDis an optional argument that indicates the dimensionality of the node data, where ND is 2 or 3.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the number of dimensions in the grid is two, as specified by the node header, then only x and y coordinates are present on each line.

The following is an example of a 2D grid:

```
(10 (1 1 2d5 1 2)(
 1.500000e-01 2.500000e-02
 1.625000e-01 1.250000e-02
   .
   .
   .
 1.750000e-01 0.000000e+00
 2.000000e-01 2.500000e-02
 1.875000e-01 1.250000e-02
 ))
```

Because the grid connectivity is composed of integers representing pointers (see Cells and Faces), using hexadecimal conserves space in the file and provides for faster file input and output. The header indices are in hexadecimal so that they match the indices in the bodies of the grid connectivity sections. The `zone-id` and `type` are also in hexadecimal for consistency.

B.3.5. Periodic Shadow Faces

Index:	18
Scheme symbol:	`xf-periodic-face`
C macro:	`XF_PERIODIC_FACE`
Status:	required only for grids with periodic boundaries

This section indicates the pairings of periodic faces on periodic boundaries. Grids without periodic boundaries do not have sections of this type. The format of the section is as follows:

```
(18 (first-index last-index periodic-zone shadow-zone)(
 f00 f01
 f10 f11
 f20 f21
 .
 .
 .
 ))
```

where

`first-index` = index of the first periodic face pair in the list

`last-index` = index of the last periodic face pair in the list

`periodic-zone` = zone ID of the periodic face zone

`shadow-zone` = zone ID of the corresponding shadow face zone

These are in hexadecimal format. The indices in the section body (`f*`) refer to the faces on each of the periodic boundaries (in hexadecimal), the indices being offsets into the list of faces for the grid.

Note

In this case, `first-index` and `last-index` do *not* refer to face indices. They refer to indices in the list of periodic pairs.

Example:

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(18 (1 2b a c) (
 12 1f
 13 21
 ad 1c2
 .
 .
 .
 ))
```

B.3.6. Cells

Index:	12
Scheme symbol:	`xf-cell`
C macro:	XF_CELL
Status:	required

The declaration section for cells is similar to that for nodes.

```
(12 (zone-id first-index last-index type element-type))
```

Again, `zone-id` is zero to indicate that it is a declaration of the total number of cells. If `last-index` is zero, then there are no cells in the grid. This is useful when the file contains only a surface mesh to alert ANSYS FLUENT that it cannot be used. In a declaration section, the `type` has a value of zero and the `element-type` is not present.

For example,

```
(12 (0 1 3e3 0))
```

It states that there are 3e3 (hexadecimal) = 995 cells in the grid. This declaration section is required and must precede the regular cell sections.

The `element-type` in a regular cell section header indicates the type of cells in the section, as follows:

`element-type`	*description*	*nodes/cell*	*faces/cell*
0	mixed		
1	triangular	3	3
2	tetrahedral	4	4
3	quadrilateral	4	4
4	hexahedral	8	6
5	pyramid	5	5
6	wedge	6	5
7	polyhedral	NN	NF

where NN and NF will vary, depending on the specific polyhedral cell.

Regular cell sections have no body, but they have a header of the same format where `first-index` and `last-index` indicate the range for the particular zone, `type` indicates whether the cell zone is an active zone (solid or fluid), or inactive zone (currently only parent cells resulting from hanging node adaption). Active zones are represented with `type`=1, while inactive zones are represented with `type`=32.

In the earlier versions of ANSYS FLUENT, a distinction was made used between solid and fluid zones. This is now determined by properties (i.e., material type).

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A `type` of zero indicates a dead zone and will be skipped by ANSYS FLUENT. If a zone is of mixed type (`element-type`=0), it will have a body that lists the `element-type` of each cell.

Example:

```
(12 (9 1 3d 0 0)(
 1 1 1 3 3 1 1 3 1
 .
 .
 .
 ))
```

Here, there are 3d (hexadecimal) = 61 cells in cell zone 9, of which the first 3 are triangles, the next 2 are quadrilaterals, and so on.

B.3.7. Faces

Index:	13
Scheme symbol:	`xf-face`
C macro:	XF_FACE
Status:	required

The format for face sections is as follows:

```
(13 (zone-id first-index last-index bc-type face-type))
```

where

`zone-id` = zone ID of the face section

`first-index` = index of the first face in the list

`last-index` = index of the last face in the list

`bc-type` = ID of the boundary condition represented by the face section

`face-type` = ID of the type(s) of face(s) in the section

The current valid boundary condition types are defined in the following table:

`bc-type`	*description*
2	interior
3	wall
4	pressure-inlet, inlet-vent, intake-fan
5	pressure-outlet, exhaust-fan, outlet-vent
7	symmetry
8	periodic-shadow
9	pressure-far-field
10	velocity-inlet
12	periodic
14	fan, porous-jump, radiator
20	mass-flow-inlet

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24	interface
31	parent (hanging node)
36	outflow
37	axis

The faces resulting from the intersection of non-conformal grids are placed in a separate face zone, where a factor of 1000 is added to the `bc-type` (e.g., 1003 is a wall zone).

The current valid face types are defined in the following table:

`face-type`	*description*	*nodes/face*
0	mixed	
2	linear	2
3	triangular	3
4	quadrilateral	4
5	polygonal	NN

where NN will vary, depending on the specific polygonal face.

A `zone-id` of zero indicates a declaration section, which provides a count of the total number of faces in the file. Such a section omits the `bc-type` and is not followed by a body with further information.

A non-zero `zone-id` indicates a regular face section, and will be followed by a body that contains information about the grid connectivity. Each line of the body will describe one face and will have the following format:

```
n0 n1 n2 c0 c1
```

where,

`n*` = defining nodes (vertices) of the face

`c*` = adjacent cells

This is the format for a 3D grid with a triangular face format. The actual number of nodes depends on the `face-type`. The order of the cell indices is important, and is determined by the right-hand rule: if you curl the fingers of your right hand in the order of the nodes, your thumb will point toward `c0`.

For 2D grids, `n2` is omitted. `c1` is determined by the cross product of two vectors, $\hat{r}$ and $\hat{k}$. The $\hat{r}$ vector extends from `n0` to `n1`, whereas the $\hat{k}$ vector has its origin at `n0` and points out of the grid plane toward the viewer. If you extend your right hand along $\hat{r}$ and curl your fingers in the direction of the angle between $\hat{r}$ and $\hat{k}$, your thumb will point along $\hat{r}\times\hat{k}$ toward `c1`.

If the face zone is of mixed type (`face-type`= 0), each line of the section body will begin with a reference to the number of nodes that make up that particular face, and has the following format:

```
x n0 n1 ... nf c0 c1
```

where,

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`x` = the number of nodes (vertices) of the face

`nf` = the final node of the face

All cells, faces, and nodes have positive indices. If a face has a cell only on one side, then either `c0` or `c1` is zero. For files containing only a surface mesh, both these values are zero.

For information on face-node connectivity for various cell types in ANSYS FLUENT, refer to *Face-Node Connectivity in ANSYS FLUENT* (p. 136).

B.3.8. Face Tree

Index:	59
Scheme symbol:	`xf-face-tree`
C macro:	XF_FACE_TREE
Status:	only for grids with hanging-node adaption

This section indicates the face hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
  (59 (face-id0 face-id1 parent-zone-id child-zone-id)
  (
  number-of-kids kid-id-0 kid-id-1 ... kid-id-n
 .
 .
 .
  ))
```

where,

`face-id0` = index of the first parent face in the section

`face-id1` = index of the last parent face in the section

`parent-zone-id` = ID of the zone containing parent faces

`child-zone-id` = ID of the zone containing children faces

`number-of-kids` = the number of children of the parent face

`kid-id-n` = the face IDs of the children

These are in hexadecimal format.

B.3.9. Cell Tree

Index:	58
Scheme symbol:	`xf-cell-tree`
C macro:	XF_CELL_TREE
Status:	only for grids with hanging-node adaption

This section indicates the cell hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
  (58 (cell-id0 cell-id1 parent-zone-id child-zone-id)
  (
  number-of-kids kid-id-0 kid-id-1 ... kid-id-n
 .
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
.
.
 ))
```

where,

`cell-id0` = index of the first parent cell in the section

`cell-id1` = index of the last parent cell in the section

`parent-zone-id` = ID of the zone containing parent cells

`child-zone-id` = ID of the zone containing children cells

`number-of-kids` = the number of children of the parent cell

`kid-id-n` = the cell IDs of the children

These are in hexadecimal format.

B.3.10. Interface Face Parents

Index:	61
Scheme symbol:	`xf-face-parents`
C macro:	XF_FACE_PARENTS
Status:	only for grids with non-conformal interfaces

This section indicates the relationship between the intersection faces and original faces. The intersection faces (children) are produced from intersecting two non-conformal surfaces (parents) and are some fraction of the original face. Each child will refer to at least one parent. The format of the section is as follows:

```
 (61 (face-id0 face-id1)
 (
 parent-id-0 parent-id-1
.
.
.
 ))
```

where,

`face-id0` = index of the first child face in the section

`face-id1` = index of the last child face in the section

`parent-id-*` = index of parent faces

These are in hexadecimal format.

If you read a non-conformal grid from ANSYS FLUENT into TGrid, TGrid will skip this section, so it will not maintain all the information necessary to preserve the non-conformal interface. When you read the grid back into ANSYS FLUENT, you will need to recreate the interface.

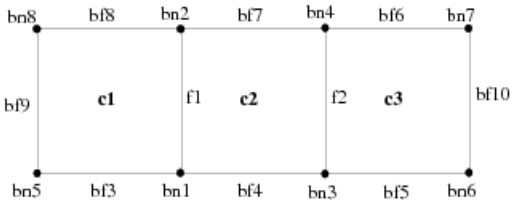
B.3.10.1. Example Files

B.3.10.1.1. Example 1

Figure B.1 (p. 2196) illustrates a simple quadrilateral mesh with no periodic boundaries or hanging nodes.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure B.1 Quadrilateral Mesh



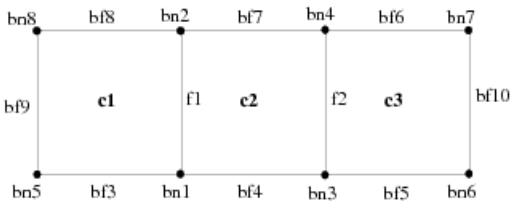
The following describes this mesh:

```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2 3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0 1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0 4 2 2 0
2 8 1 0))

(13 (5 9 9 a 2)(
8 5 1 0))

(13 (6 a a 24 2)(
6 7 3 0))

(10 (1 1 8 1 2)
(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.3.10.1.2. Example 2

Figure B.2 (p. 2196) illustrates a simple quadrilateral mesh with periodic boundaries but no hanging nodes. In this example, bf9 and bf10 are faces on the periodic zones.

Figure B.2 Quadrilateral Mesh with Periodic Boundaries



Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

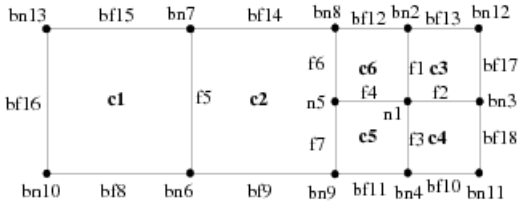
The following describes this mesh:

```
(0 "Dimensions:")
(2 2)

(0 "Grid:")

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2 3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0 1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0 4 2 2 0
2 8 1 0))

(13 (5 9 9 c 2)(
8 5 1 0))

(13 (1 a a 8 2)(
6 7 3 0))

(18 (1 1 5 1)(
9 a))

(10 (1 1 8 1 2)(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.3.10.1.3. Example 3

Figure B.3 (p. 2197) illustrates a simple quadrilateral mesh with hanging nodes.

Figure B.3 Quadrilateral Mesh with Hanging Nodes



The following describes this mesh:

```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 7 0))
(13 (0 1 16 0))
(10 (0 1 d 0 2))
```

```
(12 (7 1 6 1 3))
(12 (1 7 7 20 3))

(58 (7 7 1 7)(
4 6 5 4 3))

(13 (2 1 7 2 2)(
1 2 6 3
1 3 3 4
1 4 4 5
1 5 5 6
6 7 1 2
5 8 2 6
9 5 2 5))

(13 (3 8 b 3 2)(
a 6 1 0
6 9 2 0  4 b 4 0
9 4 5 0))

(13 (4 c f 3 2)(
2 8 6 0
c 2 3 0
8 7 2 0
7 d 1 0))

(13 (5 10 10 a 2)(
d a 1 0))

(13 (6 11 12 24 2)(
3 c 3 0
b 3 4 0))

(13 (b 13 13 1f 2)(
c 8 7 0))

(13 (a 14 14 1f 2)(
b c 7 0))

(13 (9 15 15 1f 2)(
9 b 7 0))

(13 (8 16 16 1f 2)(
9 8 2 7))

(59 (13 13 b 4)(
2 d c))

(59 (14 14 a 6)(
2 12 11))

(59 (15 15 9 3)(
2 b a))

(59 (16 16 8 2)(
2 7 6))

(10 (1 1 d 1 2)
(
2.50000000e+00 5.00000000e-01
2.50000000e+00 1.00000000e+00
3.00000000e+00 5.00000000e-01
2.50000000e+00 0.00000000e+00
2.00000000e+00 5.00000000e-01
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
```


Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.00000000e+00 1.00000000e+00))
```

B.4. Other (Non-Grid) Case Sections

The following sections store boundary conditions, material properties, and solver control settings.

B.4.1. Zone
B.4.2. Partitions

B.4.1. Zone

Index:	39 or 45
Scheme symbol:	`xf-rp-tv`
C macro:	XF_RP_TV
Status:	required

There is typically one zone section for each zone referenced by the grid. Although some grid zones may not have corresponding zone sections, there cannot be more than one zone section for each zone.

A zone section has the following form:

```
(39 (zone-id zone-type zone-name domain-id)(
(condition1 . value1)
(condition2 . value2)
(condition3 . value3)
.
.
.
 ))
```

Grid generators and other preprocessors need only provide the section header and leave the list of conditions empty, as in

```
(39 (zone-id zone-type zone-name domain-id)())
```

The empty parentheses at the end are required. The solver adds conditions as appropriate, depending on the zone type. When only `zone-id`, `zone-type`, `zone-name`, and `domain-id` are specified, the index 45 is preferred for a zone section. However, the index 39 must be used if boundary conditions are present, because any and all remaining information in a section of index 45 after `zone-id`, `zone-type`, `zone-name`, and `domain-id` will be ignored.

Here the `zone-id` is in *decimal* format. This is in contrast to the use of hexadecimal in the grid sections.

The `zone-type` is one of the following:

```
axis
 exhaust fan
 fan
 fluid
 inlet vent
 intake fan
 interface
 interior
 mass-flow-inlet
 outlet vent
 outflow
 periodic
 porous-jump
 pressure-far-field
 pressure-inlet
```

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
pressure-outlet
radiator
shadow
solid
symmetry
velocity-inlet
wall
```

The `interior`, `fan`, `porous-jump`, and `radiator` types can be assigned only to zones of faces inside the domain. The `interior` type is used for the faces within a cell zone; the others are for interior faces that form infinitely thin surfaces within the domain. ANSYS FLUENT allows the `wall` type to be assigned to face zones both on the inside and on the boundaries of the domain. Some zone types are valid only for certain types of grid components. For example, cell (element) zones can be assigned only one of the following types:

```
fluid
 solid
```

All of the other types listed above can be used only for boundary (face) zones.

The `zone-name` is a user-specified label for the zone. It must be a valid Scheme symbol [1] and is written without quotes. The rules for a valid `zone-name` (Scheme symbol) are as follows:

- The first character must be a lowercase letter [2] or a special-initial.
- Each subsequent character must be a lowercase letter, a special-initial, a digit, or a special-subsequent.

where a special-initial character is one of the following:

```
! $ % & * / : < = > ? ~ _ ^
```

and a special-subsequent is one of the following:

```
. + -
```

Examples of valid zone names are `inlet-port/cold!–`, `eggs/easy`, and `e=m*c^2`.

Some examples of zone sections produced by grid generators and preprocessors are as follows:

```
(39 (1 fluid fuel 1)())
 (39 (8 pressure-inlet pressure-inlet-8 2)())
 (39 (2 wall wing-skin 3)())
 (39 (3 symmetry mid-plane 1)())
```

The `domain-id` is an integer that appears after the zone name, associating the boundary condition with a particular phase or mixture (sometimes referred to as phase-domains and mixture-domains).

B.4.2. Partitions

Index:	40
Scheme symbol:	`xf-partition`
C macro:	`XF_PARTITION`
Status:	only for partitioned grids

[1] See Revised (4) Report on the Algorithmic Language Scheme, William Clinger and Jonathan Rees (Editors), 2 November 1991, Section 7.1.1.

[2] The Standard actually only requires that case be insignificant; the ANSYS FLUENT implementation accomplishes this by converting all uppercase input to lowercase.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section indicates each cell's partition. The format of the section is as follows:

```
(40 (zone-id first-index last-index partition-count)(
p1
p2
p3
.
.
.
pn
))
```

where,

`p1` = the partition of the cell whose ID is `first-index`

`p2` = the partition of the cell whose ID is `first-index` + 1, etc.

`pn` = the partition of the cell whose ID is `last-index`

`partition-count` = the total number of partitions

Partition IDs must be between 0 and one less than `partition-count`.

B.5. Data Sections

The following sections store iterations, residuals, and data field values.

B.5.1. Grid Size
B.5.2. Data Field
B.5.3. Residuals

B.5.1. Grid Size

Index:	33
Scheme symbol:	`xf-grid-size`
C macro:	`XF_GRID_SIZE`
Status:	optional

This section indicates the number of cells, faces, and nodes in the grid that corresponds to the data in the file. This information is used to check that the data and grid match. The format is

```
(33 (n-elements n-faces n-nodes))
```

where the integers are written in decimal.

B.5.2. Data Field

Index:	300
Scheme symbol:	`xf-rf-seg-data`
C macro:	`XF_RF_SEG_DATA`
Status:	required

This section lists a flow field solution variable for a cell or face zone. The data are stored in the same order as the cells or faces in the case file. Separate sections are written out for each variable for each face or cell zone on which the variable is stored. The format is

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(300 (sub-section-id zone-id size n-time-levels
 n-phases first-id last-id)
 ( data for cell or face with id = first-id
 data-for-cell-or-face with id = first-id+1
 ..
 data-for-cell-or-face with id = last-id
 ))
```

where `sub-section-id` is a (decimal) integer that identifies the variable field (e.g., 1 for pressure, 2 for velocity). The complete list of these is available in the header file (`xfile.h`), which is located in your installation area.

where,

`zone-id` = the ID number of the cell or face zone

`size` = the length of the variable vector

`zone-id` matches the ID used in case file. `size` is 1 for a scalar, 2 or 3 for a vector, equal to the number of species for variables defined for each species). `n-time-levels` currently are not used.

A sample data file section for the velocity field in a cell zone for a steady-state, single-phase, 2D problem is shown below:

```
(300 (2 16 2 0 0 17 100)
(8.08462024e-01 8.11823010e-02
 8.78750622e-01 3.15509699e-02
 1.06139672e+00 -3.74040119e-02
...
 1.33301604e+00 -5.04243895e-02
 6.21703446e-01 -2.46118382e-02
 4.41687912e-01 -1.27046436e-01
 1.03528820e-01 -1.01711005e-01
 ))
```

The variables that are listed in the data file depend on the models active at the time the file is written. Variables that are required by the solver based on the current model settings but are missing from the data file are set to their default values when the data file is read. Any extra variables that are present in the data file but are not relevant according to current model settings are ignored.

B.5.3. Residuals

Index:	302
Scheme symbol:	`xf-rf-scaled-residuals`
C macro:	`XF_RF_SCALED_RESIDUALS`
Status:	optional

This section lists the values of the residuals for a particular data field variable at each iteration:

```
(302 (n residual-section-id size domain-id)
(
iteration_number unscaled_residual scaling_factor
 .
 .
 .
 ))
```

where,

`n` = the number of residuals

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`size` = the length of the variable vector

`residual-section-id` = an integer (decimal) indicating the equation

`domain-id` = domain ID

`size` is 1 for a scalar, 2 or 3 for a vector, equal to the number of species for variables defined for each species. The `residual-section-id` indicates the equation for which the residual is stored in the section, according to the C constants defined in a header file (`xfile.h`) available in your installation area, as noted in *Grid Sections* (p. 2188).

The equations for which residuals are listed in the data file depend on the models active at the time the file is written. If the residual history is missing from the data file for a currently active equation, it is initialized with zeros.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix C. Nomenclature

A	Area (m ^{2}, ft ^{2})
$\vec{a}$	Acceleration (m/s ^{2}, ft/s ^{2})
a	Local speed of sound (m/s, ft/s)
c	Concentration (mass/volume, moles/volume)
C_D	Drag coefficient, defined different ways (dimensionless)
c_p, c_v	Heat capacity at constant pressure, volume (J/kg-K, Btu/lb $_m$- $^\circ$F)
d	Diameter; d_p, D_p, particle diameter (m, ft)
D_H	Hydraulic diameter (m, ft)
$_{ij}, D$	Mass diffusion coefficient (m ^{2}/s, ft ^{2}/s)
E	Total energy, activation energy (J, kJ, cal, Btu)
f	Mixture fraction (dimensionless)
$\vec{F}$	Force vector (N, lb $_f$)
F_D	Drag force (N, lb $_f$)
$\vec{g}$	Gravitational acceleration (m/s ^{2}, ft/s ^{2}); standard values = 9.80665 m/s ^{2}, 32.1740 ft/s ^{2}
Gr	Grashof number ≡ ratio of buoyancy forces to viscous forces (dimensionless)
H	Total enthalpy (energy/mass, energy/mole)
h	Heat transfer coefficient (W/m ^{2}-K, Btu/ft ^{2}-h- $^\circ$F)
h	Species enthalpy; h^0, standard state enthalpy of formation (energy/mass, energy/mole)
I	Radiation intensity (energy per area of emitting surface per unit solid angle)
J	Mass flux; diffusion flux (kg/m ^{2}-s, lb $_m$/ft ^{2}-s)
K	Equilibrium constant = forward rate constant/backward rate constant (units vary)
k	Kinetic energy per unit mass (J/kg, Btu/lb $_m$)
k	Reaction rate constant, e.g., $k_1, k_{-1}, k_{f,r}, k_{b,r}$ (units vary)
k	Thermal conductivity (W/m-K, Btu/ft-h- $^\circ$F)
k_B	Boltzmann constant (1.38×10^{-23} J/molecule-K)
k, k_c	Mass transfer coefficient (units vary); also K, K_c
ℓ, l, L	Length scale (m, cm, ft, in)
Le	Lewis number ≡ ratio of thermal diffusivity to mass diffusivity (dimensionless)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

m	Mass (g, kg, lb $_m$)
$\dot{m}$	Mass flow rate (kg/s, lb $_m$/s)
M_w	Molecular weight (kg/kmol, lb $_m$/lb $_m$mol)
M	Mach number ≡ ratio of fluid velocity magnitude to local speed of sound (dimensionless)
Nu	Nusselt number ≡ dimensionless heat transfer or mass transfer coefficient (dimensionless); usually a function of other dimensionless groups
p	Pressure (Pa, atm, mm Hg, lb $_f$/ft ^{2})
Pe	Peclet number $\equiv Re \times Pr$ for heat transfer, and $\equiv Re \times Sc$ for mass transfer (dimensionless)
Pr	Prandtl number ≡ ratio of momentum diffusivity to thermal diffusivity (dimensionless)
Q	Flow rate of enthalpy (W, Btu/h)
q	Heat flux (W/m ^{2}, Btu/ft ^{2}-h)
R	Gas-law constant (8.31447 $\times 10^3$ J/kmol-K, 1.98588 Btu/lb $_m$mol- $^\circ$F)
r	Radius (m, ft)
$\mathcal{R}$	Reaction rate (units vary)
Ra	Rayleigh number $\equiv Gr \times Pr$; measure of the strength of buoyancy-induced flow in natural (free) convection (dimensionless)
Re	Reynolds number ≡ ratio of inertial forces to viscous forces (dimensionless)
S	Total entropy (J/K, J/kmol-K, Btu/lb $_m$mol- $^\circ$F)
s	Species entropy; s^0, standard state entropy (J/kmol-K, Btu/lb $_m$mol- $^\circ$F)
Sc	Schmidt number ≡ ratio of momentum diffusivity to mass diffusivity (dimensionless)
S_{ij}	Mean rate-of-strain tensor (s $^{-1}$)
T	Temperature (K, $^\circ$C, $^\circ$R, $^\circ$F)
t	Time (s)
U	Free-stream velocity (m/s, ft/s)
u, v, w	Velocity magnitude (m/s, ft/s); also written with directional subscripts (e.g., v_x, v_y, v_z, v_r)
V	Volume (m ^{3}, ft ^{3})
$\vec{v}$	Overall velocity vector (m/s, ft/s)
We	Weber number ≡ ratio of aerodynamic forces to surface tension forces (dimensionless)
X	Mole fraction (dimensionless)
Y	Mass fraction (dimensionless)
α	Permeability, or flux per unit pressure difference (L/m ^{2}-h-atm, ft ^{3}/ft ^{2}-h-(lb $_f$/ft ^{2}))
α	Thermal diffusivity (m ^{2}/s, ft ^{2}/s)
α	Volume fraction (dimensionless)
β	Coefficient of thermal expansion (K $^{-1}$)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

γ	Porosity (dimensionless)
γ	Ratio of specific heats, c_p/c_v (dimensionless)
Δ	Change in variable, final − initial (e.g., Δp, Δt, ΔH, ΔS, ΔT)
δ	Delta function (units vary)
ε	Emissivity (dimensionless)
ε	Lennard-Jones energy parameter (J/molecule)
ε	Turbulent dissipation rate (m^2/s^3, ft^2/s^3)
ε	Void fraction (dimensionless)
η	Effectiveness factor (dimensionless)
η', η''	Rate exponents for reactants, products (dimensionless)
θ_r	Radiation temperature (K)
λ	Molecular mean free path (m, nm, ft)
λ	Wavelength (m, nm, Å, ft)
μ	Dynamic viscosity (cP, Pa-s, lb_m/ft-s)
ν	Kinematic viscosity (m^2/s, ft^2/s)
ν', ν''	Stoichiometric coefficients for reactants, products (dimensionless)
ρ	Density (kg/m^3, lb_m/ft^3)
σ	Stefan-Boltzmann constant (5.67×10^8 W/m^2-K^4)
σ	Surface tension (kg/m, dyn/cm, lb_f/ft)
σ_s	Scattering coefficient (m^{-1})
$\bar{\bar{\tau}}$	Stress tensor (Pa, lb_f/ft^2)
τ	Shear stress (Pa, lb_f/ft^2)
τ	Time scale, e.g., τ_c, τ_p, τ_c (s)
τ	Tortuosity, characteristic of pore structure (dimensionless)
ϕ	Equivalence ratio (dimensionless)
ϕ	Thiele modulus (dimensionless)
Ω	Angular velocity; Ω_{ij}, Mean rate of rotation tensor (s^{-1})
ω	Specific dissipation rate (s^{-1})
Ω, Ω'	Solid angle (degrees, radians, gradians)
Ω_D	Diffusion collision integral (dimensionless)

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Bibliography

[1] *FIELDVIEW Reference Manual, Software Release Version 10*. Intelligent Light. 2004.

[2] *KINetics for Fluent, Version 2.4*. Reaction Design, Inc., San Diego, CA,. 2009.

[3] *VdmTools Programmer Manual, Version 3.2.0*. Visual Kinematics, Inc.. 2005.

[4] *CRANIUM, Version 1.0*. Molecular Knowledge Systems, Inc., Bedford, NH. 2007.

[5] T. Ahmad, S. L. Plee, and J. P. Myers.*"Computation of Nitric Oxide and Soot Emissions from Turbulent Diffusion Flames"*. *J. of Engineering for Gas Turbines and Power*. 107. 48–53. 1985.

[6] A. A. Amsden, P. J. O'Rourke, and T. D. Butler.*"KIVA-2: A Computer Program for Chemically Reactive Flows with Sprays"*. *Technical Report LA-11560-MS, UC-96*. Los Alamos National Laboratory, Los Alamos, New Mexico. May 1989..

[7] M. J. Assael, J. P. M. Trusler, and T. F. Tsolakis.*"Thermophysical Properties of Fluids"*. *Technical report*. Imperial College Press, London. 1996.

[8] R. H. Aungier. *"A Fast, Accurate Real Gas Equation of State for Fluid Dynamic Analysis Applications"*. *Journal of Fluids Engineering*. 117. 277–281. 1995..

[9] R. Barrett, M. Berry, T. F. Chan, J. Demmel, J. Donato, J. Dongarra, V. Eijkhout, R. Pozo, C. Romine, and H. Van der Vorst. *"Templates for the Solution of Linear Systems: Building Blocks for Iterative Methods"*. *SIAM, 2nd edition*. Philadelphia, Pennsylvania. 1994.

[10] A. Bejan. *Convection Heat Transfer*. John Wiley and Sons, New York. 1984.

[11] C. T. Bowman. *"Chemistry of Gaseous Pollutant Formation and Destruction*. *Fossil Fuel Combustion*. In W. Bartok and A. F. Sarofim, editorsJ. Wiley and Sons, Canada. 1991.

[12] S. J. Brookes and J. B. Moss.*"Prediction of Soot and Thermal Radiation in Confined Turbulent Jet Diffusion Flames"*. *Combustion and Flame*. 116. 486–503. 1999.

[13] J. R. Cash and A. H. Karp.*"A variable order Runge-Kutta method for initial value problems with rapidly varying right-hand sides"*. *ACM Transactions on Mathematical Software*. 16. 201–222. 1990.

[14] T. Cebeci and P. Bradshaw.*Momentum Transfer in Boundary Layers*. Hemisphere Publishing Corporation, New York. 1977.

[15] P. J. Coelho and M. G. Carvalho.*"Modeling of Soot Formation and Oxidation in Turbulent Diffusion Flames"*. *J. of Thermophysics and Heat Transfer*. 9(4). 644–652. 1995.

[16] M. F. Cohen and D. P. Greenberg.*"The Hemi-Cube: A Radiosity Solution for Complex Environments"*. *Computer Graphics*. 19(3). 31–40. 1985.

[17] H. W. Coleman and B. K. Hodge and R. P. Taylor.*"A Re-Evaluation of Schlichting's Surface Roughness Experiment"*. *Journal of Fluids Engineering*. 106. 1984.

[18] E. H. Cuthill and J. McKee.*"Reducing Bandwidth of Sparse Symmetric Matrices"*. *In Proc. ACM 24th National Conf. New York*. 157–172. 1969.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[19] J. E. Dec. *"A Conceptual Model of DI Diesel Combustion Based on Laser Sheet Imaging"*. *SAE Technical Paper 970873*. SAE. 1997.

[20] J. Ding and D. Gidaspow. *"A Bubbling Fluidization Model Using Kinetic Theory of Granular Flow"*. *AIChE J.*. 36(4). 523–538. 1990.

[21] S. Ergun. *"Fluid Flow through Packed Columns"*. *Chem. Eng. Prog.*. 48(2). 89–94. 1952.

[22] G. M. Faeth. *"Spray Atomization and Combustion"*. *Technical Report 1986-136*. AIAA. 1986.

[23] J. E. Ffowcs-Williams and D. L. Hawkings. *"Sound Generation by Turbulence and Surfaces in Arbitrary Motion"*. *Proc. Roy. Soc. London*. A264. 321–342. 1969.

[24] E. U. Finlayson, D. K. Arasteh, C. Huizenga, M. D. Rubin, and M. S. Reilly. *"WINDOW 4.0: Documentation of Calculation Procedures"*. *Technical Report Publication LBL-33943/ TA-309*. Lawrence Berkeley Laboratory, Energy and Environmental Division, Berkeley, CA 94720. 1993.

[25] A. Ghobadian and S. A. Vasquez. *"A General Purpose Implicit Coupled Algorithm for the Solution of Eulerian Multiphase Transport Equation"*. *In International Conference on Multiphase Flow, Leipzig, Germany*. 2007.

[26] N. E. Gibbs, W. G. Poole, Jr., and P. K. Stockmeyer. *"An algorithm for reducing the bandwidth and profile of a sparse matrix"*. *SIAM J. Numer. Anal.*. 13. 236–250. 1976.

[27] M. B. Giles. *"Non-Reflecting Boundary Conditions for the Euler Equations"*. *Technical Report 88-1*. Computational Fluid Dynamics Laboratory, Massachusetts Institute of Technology, Cambridge, MA,. 1988.

[28] J. Gottgens, F. Mauss, and N. Peters. *"Analytic Approximations of Burning Velocities and Flame Thicknesses of Lean Hydrogen, Methane, Ethylene, Ethane, Acetylene and Propane Flames"*. *In Twenty-Fourth Symposium (Int.) on Combustion, Pittsburgh*. 129–135. 1992.

[29] P. M. Gresho, R. L. Lee, and R. L. Sani. *"On the Time-Dependent Solution of the Incompressible Navier-Stokes Equations in Two and Three Dimensions"*. *In Recent Advances in Numerical Methods in Fluids*. Pineridge Press, Swansea, U. K.. 1980.

[30] R. J. Hall, M. D. Smooke, and M. B. Colket. *Physical and Chemical Aspects of Combustion*. Gordon and Breach. 1997.

[31] R. A. W. M. Henkes and C. J. Hoogendoorn. *"Scaling of the Turbulent Natural Convection Flow in a Heated Square Cavity"*. *Trans. of the ASME*. 116. 400–408. May 1994.

[32] J. B. Heywood. *Internal Combustion Engine Fundamentals*. McGraw-Hill, New York, rev. edition. 1988.

[33] C. Hirsch. *Numerical Computation of Internal and External Flows: Computational Methods for Inviscid and Viscous Flows, volume 2*. John Wiley & Sons, New York. 1984.

[34] J. O. Hirschfelder, C. F. Curtiss, and R. B. Bird. *Molecular Theory of Gases and Liquids*. John Wiley & Sons, New York. 1954.

[35] K. Hsu and A. Jemcov. *"Numerical investigation of detonation in premixed hydrogen-air mixture - assessment of simplified chemical mechanisms"*. *Technical Report AIAA-2000-2478*. American Institute of Aeronautics and Astronautics, Fluids 2000 Conference and Exhibit, Denver, CO. June 2000.

[36] G. W. Jackson and D. F. James. *"The Permeability of Fibrous Porous Media"*. *Canadian Journal of Chem. Eng.*. 64(3). 364–374. June 1986.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[37] A. Jemcov and J. P. Maruszewski. *"Shape Optimization Based on Downhill Simplex Optimizer and Free-Form Deformation in General Purpose CFD Code"*. *17th Annual Conference of CFD Society of Canada*. Ottawa, Ontario, Canada. 2009.

[38] P. C. Johnson and R. Jackson. *"Frictional-Collisional Constitutive Relations for Granular Materials, with Application to Plane Shearing"*. *J. Fluid Mech.*. 176. 67–93. 1987.

[39] G. Karypis and V. Kumar. *METIS - A Software Package for Partitioning Unstructured Graphs, Partitioning Meshes, and Computing Fill-Reducing Orderings of Sparse Matrices, Version 3.0. Manual*. University of Minnesota and Army HPC Research Center. 1997.

[40] R. J. Kee, F. M. Rupley, and J. A. Miller. *"SURFACE CHEMKIN: A Software Package for the Analysis of Heterogeneous Chemical Kinetics at a Solid-Surface - Gas-Phase Interface"*. *Technical Report SAND 96-8217*. Sandia National Labs. 2000.

[41] R. J. Kee, F. M. Rupley, J. A. Miller, M. E. Coltrin, J. F. Grcar, E. Meeks, H. K. Moffat, A. E. Lutz, G. Dixon-Lewis, M. D. Smooke,J. Warnatz,G. H.Evans,R. S.Larson,R. E.Mitchell,L. R. Petzold, W. C. Reynolds, M. Caracotsios, W. E. Stewart, P. Glarborg, C. Wang, O. Adigun, W. G. Houf, C. P. Chou, S. F. Miller, P. Ho, and D. J. Young. *"CHEMKIN v. 4.0. Technical Report San Diego, CA*. Reaction Design, Inc.. 2004.

[42] K. K. Y. Kuo. *Principles of Combustion*. John Wiley and Sons, New York. 1986.

[43] B. E. Launder and D. B. Spalding. *"The Numerical Computation of Turbulent Flow"*. *Computer Methods in Applied Mechanics and Engineering*. 3. 269–289. 1974.

[44] M. J. Lighthill. *"On Sound Generated Aerodynamically"*. *Proc. Roy. Soc. London*. A211. 564–587. 1952.

[45] R. P. Lindsted, F. C. Lockwood, and M. A. Selim. *"Rate Constants Based on Partial Equilibrium Assumptions"*. *Internal Report No. TF/95/3, Technical report*. Imperial College, London, UK. 1995.

[46] M. S. Liou. *"A sequel to AUSM: AUSM+"*. *Journal of Computational Physics*. 129. 364–382. 1996.

[47] F. C. Lockwood, S. M. A. Rizvi, and N. G. Shah. *"Comparative Predictive Experience of Coal Firing"*. *In Proceedings Inst. Mechanical Engns.*. 200. 79–87. 1986.

[48] J. Y. Luo, R. I. Issa, and A. D. Gosman. *"Prediction of Impeller-Induced Flows in Mixing Vessels Using Multiple Frames of Reference"*. *In I ChemE Symposium Series*. 136. 549–556. 1994.

[49] B. F. Magnussen and B. H. Hjertager. *"On mathematical models of turbulent combustion with special emphasis on soot formation and combustion"*. *In 16th Symp. (Int'l.) on Combustion*. The Combustion Institute. 1976.

[50] M. Manninen, V. Taivassalo, and S. Kallio. *"On the mixture model for multiphase flow"*. *VTT Publications 288*. Technical Research Centre of Finland. 1996.

[51] B. J. McBride, S. Gordon, and M. A. Reno. *"Coefficients for Calculating Thermodynamic and Transport Properties of Individual Species"*. *NASA TM-4513*. October 1993.

[52] H. A. McGee. *Molecular Engineering*. McGraw-Hill, New York. 1991.

[53] F. R. Menter and Y. Egorov. *"Re-visiting the Turbulent Scale Equation"*. *Proc. IUTAM Symp. One Hundred Years of Boundary Layer Research, Göttingen.*. Springer. 2004.

[54] F. R. Menter and Y. Egorov. *"A Scale-Adaptive Simulation Model Using Two-Equation Models"*. *AIAA Paper-2005-1095*. Reno NV. 2005.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[55] H. J. Merk. *"The Macroscopic Equations for Simultaneous Heat and Mass Transfer in Isotropic, Continuous and Closed Systems"*. *Appl. Sci. Res.*. 8. 73–99. 1958.

[56] R. Merz, J. Kruckels, J. Mayer, and H. Stetter.*"Computation of Three Dimensional Viscous Transonic Turbine Stage Flow Including Tip Clearance Effects"*. *ASME 95-GT-76*. 1995.

[57] P. J. O'Rourke. *"Collective Drop Effects on Vaporizing Liquid Sprays"*. *PhD thesis*. Princeton University, Princeton, New Jersey. 1981.

[58] M. Østberg, P. Glarborg, A. Jensen, J. E. Johnsson, L. S. Pedersen, and K. Dam-Johansen.*"A model of the coal reburning process"*. *In Proc. Combust Inst.*. 27. 3027–3035. 1998.

[59] I. Owczarek and K. Blazej. *"Recommended Critical Temperatures. Part I. Aliphatic Hydrocarbons"*. *J. Phys. Chem.*. 32. 1411–1427. 2003.

[60] I. Owczarek and K. Blazej. *"Recommended Critical Pressures. Part I. Aliphatic Hydrocarbons"*. *J. Phys. Chem.*. 35. 1461–1474. 2006.

[61] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Hemisphere, Washington, DC. 1980.

[62] S. V. Patankar, C. H. Liu, and E. M. Sparrow.*"Fully Developed Flow and Heat Transfer in Ducts Having Streamwise-Periodic Variations of Cross-Sectional Area"*. *ASME J. of Heat Transfer*. 99. 180–186. 1977.

[63] R. H. Perry, D. W. Gree, and J. O. Maloney.*Perry's Chemical Engineers' Handbook*. McGraw-Hill, New York, 6th edition. 1984.

[64] A. A. F. Peters and R. Weber.*"Mathematical Modeling of a 2.25 MWt Swirling Natural Gas Flame. Part 1: Eddy Break-up Concept for Turbulent Combustion; Probability Density Function Approach for Nitric Oxide Formation"*. *Combustion Science and Technology*. 110. 67–101. 1995.

[65] T. J. Poinsot and S. K. Lele. *"Boundary Conditions for Direct Simulation of Compressible Viscous Flows"*. *Journal of Computational Physics*. 101. 104–129. 1992.

[66] T. J. Poinsot and L. Selle. *"Actual Impedence of Nonreflecting Boundary Conditions: Implications for Computation of Resonators"*. *AIAA Journal*. 42(5). 958–964. May 2004.

[67] B.E. Poling, J.M. Prausnitz, and J.P. OConnell. *The properties of Gases and Liquids*. McGraw-Hill, International Edition, 5th edition. 2007.

[68] S. B. Pope. *ISAT-CK (Version 3.0) User's Guide and Reference Manual*,. 2000.

[69] W. C. Reynolds. *Thermodynamic Properties in SI: Graphs, Tables, and Computational Equations for 40 Substances*. Department of Mechanical Engineering, Stanford University. 1979.

[70] J. W. Rose and J. R. Cooper.*Technical Data on Fuel*. Scottish Academic Press, Edinburgh. 1977.

[71] J. W. Rose and J. R. Cooper, editors.*Technical Data on Fuels*. Wiley, 7th edition. 1977.

[72] A. Saxer. *"A Numerical Analysis of a 3D Inviscid Stator/Rotor Interaction Using Non-Reflecting Boundary Conditions"*. *PhD thesis*. Massachusetts Institute of Technology, Cambridge, Massachusetts. March 1992.

[73] S. Sarkar and L. Balakrishnan. *"Application of a Reynolds-Stress Turbulence Model to the Compressible Shear Layer"*. ICASE Report 90-18*NASA CR 182002*. 1990.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[74] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. Kh. Strelets, and A. Travin. *"A New Version of Detached-eddy Simulation, Resistant to Ambiguous Grid Densities "*. *Theoretical and Computational Fluid Dynamics*. Springer Berlin / Heidelberg. Volume 20, Number 3. July, 2006.

[75] H. Schlichting. *Boundary-Layer Theory*. McGraw-Hill, New York. 1979.

[76] H. Schlichting and K. Gersten. *Grenzschicht-Theorie*. Springer-Verlag, Berlin, Heidelberg, New York. 1997.

[77] A. A. Shabana. *Computational Dynamics*. John Wiley and Sons, New York. 1994.

[78] G. M. Shroff and H. B. Keller. *"Stabilization of unstable procedures: The recursive projection method"*. *SIAM J. Numer. Anal.*. 30(4). 1099–1120. 1993.

[79] M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin.*"A Hybrid RANS-LES Approach With Delayed-DES and Wall-Modelled LES Capabilities"*. *International Journal of Heat and Fluid Flow*. 29:6. December 2008. 1638-1649.

[80] Y. R. Sivathanu and G. M. Faeth. *"Generalized State Relationships for Scalar Properties in Non-Premixed Hydrocarbon/Air Flames"*. *Combustion and Flame*. 82. 211–230. 1990.

[81] I. B.Sladkov. *"Physicochemical Properties of Methyl-and Ethylhalosilanes"*. *Russian Journal of Applied Chemistry*. 74. 1801–1805. 2001.

[82] A. L. Smith. *"Family Plots for Evaluating Physical Properties of Organosilicon Compounds"*. *AIChE J.*. 40. 373–377. 1994.

[83] P. L. Smith Jr. and M. Van Winkle.*"Discharge coefficients through perforated plates at reynolds numbers of 400 to 3,000"*. *AIChE J.*. 4(3). 266–268. 1958.

[84] D. O. Snyder, E. K. Koutsavdis, and J. S. R. Anttonen.*"Transonic store separation using unstructured cfd with dynamic meshing"*. *Technical Report AIAA-2003-3913*. American Institute of Aeronautics and Astronautics, 33th AIAA Fluid Dynamics Conference and Exhibit. 2003.

[85] G. R. Somayajulu. *"Estimation Procedures for Critical Constants"*. *J. Chem. Eng. Data*. 34. 106–120. 1989.

[86] K. Stueben. *"Introduction to Algebraic Multigrid"*. *Multigrid*. C. W. Oosterlee U. Trottenberg and A. Schuller, editorsAcademic Press, New York. 413–532. 2001.

[87] K. Sutton and P. A. Gnoffo. *"Multi-component Diffusion with Application to Computational Aerothermodynamics"*. *AIAA Paper 98-2575*. AIAA. 1998.

[88] M. Syamlal, W. Rogers, and T. J. O'Brien.*MFIX Documentation: Volume 1, Theory Guide*. DOE/METC-9411004, NTIS/DE9400087National Technical Information Service, Springfield, VA. 1993.

[89] W. TabakoffandT. Wakeman. *"Measured particle rebound characteristics useful for erosion prediction"*. *ASME paper 82-GT-170*. 1982.

[90] L. Talbot et al. *"Thermophoresis of Particles in a Heated Boundary Layer"*. *J. Fluid Mech.*. 101(4). 737–758. 1980.

[91] W. Tang and K. Brezinsky. *"Chemical kinetic simulations behind reflected shock waves"*. *International Journal of Chemical Kinetics*. 38(2). 75–97. February 2006.

[92] R. I. Tanner. *Engineering Rheology*. Clarendon Press, Oxford, rev. edition. 1988.

[93] R. Taylor and R. Krishna.*Multicomponent Mass Transfer*. Wiley, New York. 1993.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[94] C. Temperton. *"Implementation of a Self-Sorting In-Place Prime Factor FFT Algorithm"*. *Journal of Computational Physics*. 58. 283–299. 1985.

[95] P. A. Tesner, T. D. Snegiriova, and V. G. Knorre.*"Kinetics of Dispersed Carbon Formation"*. *Combustion and Flame*. 17. 253–260. 1971.

[96] K. W.Thompson. *"Time Dependent Boundary Conditions for Hyperbolic Systems"*. *Journal of Computational Physics*. 68. 1–24. 1987.

[97] K. W.Thompson. *"Time Dependent Boundary Conditions for Hyperbolic Systems II"*. *Journal of Computational Physics*. 89. 439–461. 1990.

[98] H. Wagner. *"Soot Formation in Combustion"*. *In 17th Symp. (Int'l.) on Combustion*. The Combustion Institute. 3–19. 1979.

[99] J. M. Weiss. *"Calculations of Reacting Flowfield Involving Stiff Chemical Kinetics"*. *AIAA-99-3369*. January 1999.

[100] J. M. Weiss, J. P. Maruszewski, and W. A. Smith.*"Implicit Solution of the Navier-Stokes Equations on Unstructured Meshes"*. *Technical Report AIAA-97-2103, 13th AIAA CFD Conference, Snowmass, CO*. July 1997.

[101] C. Westbrook and F. Dryer.*"Simplified Reaction Mechanisms for Oxidation of Hydrocarbon Fuels in Flames"*. *Comb. Sci. Tech*. 27. 31–43. 1981.

[102] D. C. Wilcox. *Turbulence Modeling for CFD*. DCW Industries, Inc., La Canada, California. 1998.

[103] V. Zimont, W. Polifke, M. Bettelini, and W. Weisenstein.*"An Efficient Computational Model for Premixed Turbulent Combustion at High Reynolds Numbers Based on a Turbulent Flame Speed Closure"*. *J. of Gas Turbines Power*. 120. 526–532. 1998.

[104] G. Soave. *"Equilibrium Constants from a modified Redlich-Kwong equation of State"*. *Chemical Engineering Science*. 27. 1197. 1972.

[105] D.Y. Peng and D.B. Robinson. *"A New Two-Constant Equation of State"*. *Industrial and Engineering Chemistry: Fundamentals*. 15. 59–64. 1976.

[106] O. Redlich and J.N.S. Kwong. *"On the Thermodynamics of Solutions. An Equation of State. Fugacities of Gaseous Solutions"*. *Chem. Rev.*. 44. 233. 1949.

[107] V. Torczon. *"On the Convergence of the Multidirectional Search Algorithm"*. *SIAM J. Optimization*. 1. 1. 123–145. 1991.

[108] W. H. Press, S. A. Teukolsky, W. T. Vetterling, and B. P. Flannery. *Numerical Recipes: The Art of Scientific Computing, Third Edition*. Cambridge University Press, Cambridge, England. 2007.

[109] H. H. Rosenbrock. *"An Automatic Method for Finding the Greatest or Least Value of a Function"*. *Computer Journal*. 3. 3. 175–184. 1960.

[110] T. G. Kolda, R. M. Lewis, and V. Torczon. *"Optimization by Direct Search: New Perspectives on Some Classical and Modern Methods"*. *SIAM Review*. 45. 3. 385–482. 2003.

[111] K. I. M. McKinnon. *"Convergence of the Nelder-Mead Simplex Method a Nonstationary Point "*. *SIAM J. Optimization*. 9. 1. 148–158. 1998.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[112] J. A. Nelder and R. Mead. *"A Simplex Method for Function Minimization"*. *Computer Journal*. 7. 4. 308–313. 1965.

[113] A. Jemcov and J. P. Maruszewski. *"Shape Optimization Based on Downhill Simplex Optimizer and Free-Form Deformation in General Purpose CFD Code"*. *17th Annual Conference of CFD Society of Canada*. Ottawa, Ontario, Canada. 2009.

[114] B. Koncar, I. Kljenak and R. Mavko. *"Modeling of Local Two-Phase Flow Parameters in Upward Subcooled Flow Boiling at Low Pressure"*. *International Journal of Heat and Mass Transfer*. 47. 1499–1513. 2004.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Index

Symbols

.cxlayout file, 1432
.fluent files, 123
1D Simulation Library dialog box, 2116
2.5D model, 583
2.5D surface remeshing method, 582

A

ABAQUS, 84
ABAQUS files
  exporting, 84
    after a calculation, 87
    during a transient calculation, 98
  importing, 77
  mapping data with, 113
absolute angular coordinate, 1549
absolute pressure, 449, 1547
absolute reference frame, 1021
absolute velocity, 517, 520, 524, 528, 533, 547, 1241–1242, 1246
absolute velocity formulation, 518
absorbed IR solar flux, 1548
absorbed radiation flux, 1548
absorbed visible solar flux, 1548
absorption coefficient, 436, 723, 1548
  composition-dependent, 436
  constant, 436
  effect of particles on, 437
  effect of soot on, 437, 974, 976, 981
  non-gray radiation, 437
  WSGGM, 436
accretion, 1017, 1054, 1553
  reporting, 1087
accuracy, 146
  first-order, 1211
  first-to-higher order, 1211
  second-order, 1211
acentric factor, 1548
acoustic power, 1548
acoustic power level, 1548
Acoustic Receivers dialog box, 998, 1734
acoustic signals, 988
Acoustic Signals dialog box, 1000, 1963
Acoustic Sources dialog box, 995, 1733
acoustics model, 987
  acoustic analogy, 988
  broadband noise, 988, 1003
    postprocessing, 1004
  CGNS export, 994
  computing sound pressure data, 1000
  direct method, 987
  FW-H, 989
    postprocessing, 1000
    solving, 1000
  integral method, 988
  quadrupoles, 988
  receivers, 998
  saving source data, 997
  source surfaces, 995
  SYSNOISE export, 993
  time step, 999
  writing
    acoustic signals, 1000
    data files, 992, 997
Acoustics Model dialog box, 990, 1731
Activate Cell Zones dialog box, 197, 2083
activating
  cells in parallel, 197
  zones, 197
active cell partition, 1548, 1611
Adapt/Anisotropic..., 1340
Adapt/Anisotropic... menu, 1339
Adapt/Boundary..., 1329
Adapt/Boundary... menu, 1328
Adapt/Controls..., 1347
Adapt/Controls... menu, 1347
Adapt/Display Options..., 1346
Adapt/Display Options... menu, 1346
Adapt/Geometry, 1341
Adapt/Geometry... menu, 1341
Adapt/Gradient..., 1331
Adapt/Gradient... menu, 1331
Adapt/Iso-Value..., 1335
Adapt/Iso-Value... menu, 1335
Adapt/Manage..., 1343
Adapt/Manage... menu, 1343
Adapt/Region..., 1336
Adapt/Region... menu, 1336
Adapt/Smooth/Swap..., 1349
Adapt/Smooth/Swap... menu, 1349
Adapt/Volume..., 1337
Adapt/Volume... menu, 1337
Adapt/Yplus/Ystar..., 1338
Adapt/Yplus/Ystar... menu, 1338
adaption, 147, 1325
  anisotropic, 1339
  boundary, 1328
  curvature, 1549
  deleting registers, 1345
  display options, 1346
  displaying registers, 1346

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamic gradient, 1333
eligibility, 1327
example, 1326
function, 1548
geometry-based, 160, 1341
gradient, 1331
guidelines, 1327
isovalue, 1335, 1548
limiting, 1347
manipulating registers, 1343
modifying registers, 1345
region, 1336
space gradient, 1548
volume, 1337
y+, y*, 1338
Adaption Display Options dialog box, 1346, 2134
Adaptive Time Step Settings dialog box, 1960
adaptive time stepping, 1262, 1264
adding text, 1427
Advanced Solution Controls dialog box, 1220, 1225, 1227, 1230–1232, 1237, 1906
aerodynamic noise, 987
agglomerating cells, 178
skewness-based approach, 182
aggregative multigrid solver, 1228
alphanumeric reporting, 1509
angle of internal friction, 1157
angular coordinate, 1549
angular discretization, 720
angular velocity, 268, 516, 524, 528
coordinate-system constraints, 515
Animate dialog box, 1454, 1986
animation, 1296, 1444, 1454
all, 1425
automatic, 1296
example, 1450–1451
of pathlines, 1070, 1402
playback, 1455
restrictions, 1458
saving, 1456
solution, 1296
animation options, 1425
Animation Recording Options dialog box, 2153
Animation Sequence dialog box, 1296, 1953
anisotropic
adaption, 1339
Anisotropic Adaption dialog box, 1340, 2127
Anisotropic Conductivity dialog box, 1762
anisotropic diffusivity, 428, 485
anisotropic thermal conductivity, 422
Anisotropic UDS Diffusivity dialog box, 429
Annotate dialog box, 1415, 2010
annotated text, 1415, 1427
Ansoft, 109
ANSYS CFD-Post, 104
exporting, 104
exporting data to, 84, 89
writing data files for, 121
ANSYS CFX files
importing, 77
ANSYS CFX files, 154
ANSYS FIDAP files, importing, 79
ANSYS FIDAP neutral files, 157
ANSYS FLUENT model compatibility, 2183
append case file, 157
append data file, 157
application window, 1428
ARIES, 153
Arrhenius reaction rate, 812
ASCII file export, 84
ASCII files
exporting, 84, 89
during a transient calculation, 98
aspect ratio, 142, 145, 147
asynchronous, 63, 69
Aungier-Redlick-Kwong Real Gas Model, 454
Auto Partition Mesh dialog box, 1604, 2160
autoignition
model, 933
Autoignition Model dialog box, 933, 1703
automatic export definitions
for particle history data, 102
for solution data, 100
Automatic Export dialog box, 100, 1946
automatic file exporting, 98
automatic file numbering, 61
automatic file saving, 65, 548, 627, 1259
automatic mesh display, 2, 36
Automatic Particle History Data Export dialog box, 102, 1949
automatic partitioning, 1604
Automatic Solution Initialization and Case Modification dialog box, 1952
automotive underhood simulations, 193, 715
Autosave Case During Mesh Motion Preview dialog box, 1897
Autosave dialog box, 65, 1943
autosave, maximum number of files, 65
averaged values, 1285, 1288
AVS, 84
AVS files, exporting
during a transient calculation, 98
AVS files, exporting, 84, 89
axes, 1480

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

attributes, 1480
Axes dialog box, 1480, 2022
axial coordinate, 1549
axial force, 1491
axial pull velocity, 1549
axial velocity, 267, 1549
axis boundary condition, 323
Axis dialog box , 1814
axis of rotation, 202, 217, 219, 515–516, 524, 528
axisymmetric cases, 143, 172, 202, 323, 500, 1509
axisymmetric flow
in multiple reference frames, 522
modeling with swirl or rotation, 500
rotation axis, 515
axisymmetric mesh setup, 1633
axisymmetric swirl flows, 259, 267, 498, 500
inputs for, 501
postprocessing for, 503
solution setup for, 502
solution stability of, 503
solution strategies for, 501

B

backflow, 282, 284
background color, 120
batch execution, 14, 1288
options, 16
Batch Options dialog box, 2073
Batch Options... dialog box, 16
beam irradiation flux, 1549
Bernoulli equation, 263
Bernstein polynomials, 631
Biaxial Conductivity dialog box, 423, 1759
biaxial thermal conductivity, 423
binary diffusivity, 1060
binary files, 59
Bingham plastics, 417
black body emission factor, 706, 722
black body temperature, 724
Blake-Kozeny equation, 232
blending
first-to-higher order, 1211
blocking surfaces, 715
blowers, 535
body forces, 1099, 1212
for multiphase flow, 1099
implicit treatment of, 1099
boiling
point, 1060
booleans , 50
boundary adaption, 1328
Boundary Adaption dialog box, 1329, 2117
boundary cell distance, 1329, 1549
boundary conditions, 205
axis, 323
case check, 1310
centerline, 323
changing zones, 207
compressible flow, 507
copying, 208
coupling with GT-Power, 379
coupling with WAVE, 381
discrete ordinates (DO) radiation model
walls, 725–726, 728
discrete phase, 1050
exhaust fan, 249, 299
fan, 324
fixing values in cell zones, 241
flow inlets and exits, 249
heat exchanger, 330
ill-posed, 294
inlet, 244, 264
inlet vent, 249, 278
intake fan, 249, 280
listing, 1525
mass flow inlet, 249, 271
mass, momentum, energy sources, 244
moving reference frame, 517
moving zones
fluids, 217
solids, 220
multiphase flow, 1107
multiple reference frames, 524
non-premixed combustion model, 883
non-reflecting, 339
non-uniform inputs for, 210
NOx, 958
outflow, 249, 293
outlet vent, 249, 297
overview, 205
periodic, 764
porous jump, 337
solar load model, 756
porous media, 221
premixed turbulent combustion, 899
pressure far-field, 249, 290
calculation setup for, 293
inputs, 291
pressure inlet, 249, 255
pressure outlets, 249, 282
density-based solver, 286
optional inputs for, 288
pressure-based solver, 286
radiant, 318

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

radiation
black body temperature, 724
defining, 724
emissivity, 724
inlets and exits, 724
S2S model, 725
walls, 725
radiator, 330
reading, 72
rotating flow, 501
saving, 72
setting, 208
transient, 376
sliding meshes, 541
soot, 984
source term input, 244
SOx, 970
species, 314, 822
surface reaction, 315
swirling flow, 501
symmetry, 319
thermal, 318, 691
at walls, 309
transient, 376
turbulence, 678
input methods, 250
types of, 205
velocity inlet, 249, 264
writing, 72
Boundary Conditions task page, 206, 1812
boundary layers, 146, 500
converting to polyhedra, 178
deformation, 561
boundary mesh file, 72
boundary normal distance, 1330, 1550
boundary profiles
reading and writing files, 70
units of, 125
boundary volume distance, 1550
boundary zones, 207
changing name of, 209
changing type of, 207
bounded plane, 1364
Bounding Frame dialog box, 1452, 2000
bounding frames, 1452
Boussinesq model, 404, 697
limitations, 698
branch cuts, 191
breakage kernel, 1150
Brownian force, 1016
bubbles, 1007
bulk species, 807–808
buoyancy-driven flows, 697
boundary conditions, 255
inputs for, 698
operating density, 700
postprocessing, 701
solution procedures, 700
solution strategies, 700
burnout stoichiometric ratio, 858, 1061
burnt mixture
species concentration, 894, 903
buttons, 28

C

C-equation, 896
C-type meshes, 191
Calculation Activities task page, 98, 1942
calculations, 1247, 1249, 1255
convergence of, 1232, 1317
executing commands during, 1288
in batch mode, 14
interrupting, 1249
pseudo transient, 1249
stability of, 1317–1318
steady-state, 1247
time-dependent, 1255
Camera Parameters dialog box, 1434, 1438, 2005
canceling
a display operation, 37
a task, 32
captions, 1413
carpet plot, 1390
Carreau model, 415
Carreau Model dialog box, 416, 1756
carrier, 830
Cartesian coordinate system, for boundary condition inputs, 259
Cartesian velocities, 1531
for boundary condition inputs, 266
case check, 1304
boundary conditions, 1310
grid, 1306
material properties, 1313
model, 1308
solver, 1314
Case Check dialog box, 1959
case files
automatic saving of, 65, 548, 627, 1259
FLUENT 4, 83, 156
FLUENT UNS, 70
format, 2187
RAMPANT, 70, 156
reading, 63–65

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

writing, 63–65, 183
case modification, 1292, 1952
cavitation model
noncondensable gases, 1153
surface tension coefficient, 1153
vaporization pressure, 1153
cell
agglomeration, 178
skewness-based approach, 182
aspect ratio, 142, 145, 147
children, 1550
element type, 1550
equiangular skewness, 1550
equivolume skewness, 1550
orthogonal quality, 144, 147, 1562
partition, 1551
refine level, 1551
Reynolds number, 1551
skewness, 142, 147
equiangular, 1550
skewness equivolume, 1550
surface area, 1551
types, 129
values, 1529
for contours, 1392
for pathlines, 1409
for XY plots, 1472
volume, 148, 1551
2D, 1551
change, 148, 1551
volume derivative, 1551
volume error, 1551, 1555
wall distance, 1551
warpage, 1551
zone type, 1552
zones, 1552
activating, 197
deactivating, 196
deleting, 196
modifying, 195
reordering, 199
replacing, 195
separating, 185, 188
cell zone conditions, 205
changing zones, 207
copying, 208
overview, 205
sliding meshes, 541
species, 822
types of, 205
Cell Zone Conditions task page, 1794
cell zone remeshing method, 571
cell zones
changing type of, 207
requirements
dynamic meshes, 571
center of pressure, 1515
report, 1515
centering the display, 1435
centerline boundary conditions, 323
CFL condition, 1222
CGNS files, 84
exporting, 84, 90
during a transient calculation, 98
importing, 78
character strings , 51
characteristic length, 448
check box, 28
checking
case setup, 1304
checking meshes, 172
checkpointing, 17
using LSF, 17
using SGE, 17
chemical mechanism
dimension reduction, 924, 927
chemical mechanism files, 84, 827, 848
chemical reaction
equilibrium chemistry, 839, 843
non-equilibrium chemistry
rich flammability limit, 847
reversible, 819
chemical vapor deposition, 829
chemistry agglomeration, 927
chemistry model, detailed, 985
CHEMKIN files, 84, 827, 848, 985
CHEMKIN Mechanism Import dialog box, 827, 832, 2055
Chemkin Mechanism Import dialog box, 1696
circumferential average, 533, 547, 1475
cleaning up processes, 18
cleanup-script, 18
cloud tracking, 1045
clustering, 708
display, 738
S2S radiation model, 711
defining automatically, 712
defining manually, 711
Coal Calculator dialog box, 1693
coal combustion, 844, 858
carbon conversion, 1073
char burn out, 1062
char burnout, 1061
devolatilization, 1061
discrete phase, 1025

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

empirical fuel definition in, 846, 854
fuel composition, 855
heating value, 854, 856
in non-premixed combustion model, 853, 855
models for, 1061
water content, 856, 858, 1042
coalescence kernel, 1150
coarse grid levels, 1225, 1232
coarsening
parameters, 1229
pathlines, 1404
cold flow, 823
color scheme for graphics windows, 2
color, background, 120
colormap, 1417
alignment, 1415
custom, 1421
disabling, 1414
labels, 1419
Colormap dialog box, 1417, 2007
Colormap Editor dialog box, 1421, 2008
colors, 1447, 2024
contour, 1387, 1496–1497
for annotation, 1415
mesh, 1383
pathline, 1404
vector, 1397–1398
combustible fraction, 858, 1061
combustion, 823
applications, 485
coal, 844, 855
empirical fuel definition, 846
equilibrium chemistry, 839, 843
liquid fuel, 844
model compatibility, 2183
non-equilibrium chemistry
rich flammability limit, 847
partially premixed, 905
pollutant formation, 862, 943
premixed, 893
solution techniques, 1319
sulfur in, 853
command files, 1288
command line history , 48
command line options, 10
Linux/UNIX, 10
parallel
Linux/UNIX, 1599
Windows, 1595
Windows, 10
command macros, 1290
commands
abbreviations, 48
aliases, 49
scheme, 49
compass optimizer, 632
Compiled UDFs dialog box, 2109
composition PDF transport
Eulerian
boundary conditions, 916
enabling, 915
Lagrangian
enabling, 913
composition PDF transport model
model compatibility, 2183
particle tracking, 921
reporting options, 920–921
compressed files, 59
compressed row format (CRF), 717
converting to, 720
compressibility factor, 1552
compressible flows, 503, 1207
boundary conditions for, 507
calculations at inlet pressure boundaries, 264, 1311
equations for, 506
floating operating pressure
enabling of, 508
limitations, 508
monitoring, 509
overview, 507
setting initial value, 509
theory, 508
gas law equation, 506
higher-order density interpolation, 1213
inputs for, 406, 506
model usage of, 505
physics of, 505
pressure interpolation, 1212
reporting results of, 510
solution strategies, 509
compressive scheme
phase localized, 1141
computational expense, 142, 1325
compute cluster package (CCP), 1597
compute nodes, 1585
connectivity information for, 1625
latency and bandwidth information for, 1628
computing
sound pressure data, 1000
view factors, 709
limitation, 717
concentration, discrete phase, 1553
concrete, 418
conduction, 691

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

shell, 313
conductive heat transfer
modeling, 691
conical mesh interface, 538
conservation equations
source term additions to, 244
consistency factor, 417
console, 26
contact angle, 1116–1117, 1139
contact resistivity, 1552
continuous casting
inputs for, 1204
contour plotting, 1386
for turbomachinery, 1496–1497
Contours dialog box, 1387, 1967
control points, 631, 634
convective flux, 485
convective heat transfer
modeling, 691
conventions used in this guide, lxiii
convergence, 700, 823, 1268, 1317
acceleration of, 1225–1226, 1232, 1239
criteria
choosing, 1274
modifying, 1276
judging, 1269, 1278, 1317
conversion factors for units, 128
Convert Skewed Cells dialog box, 2076
converting meshes, 177
coordinates, 1565, 1579
constraints for moving reference frames, 515
Copy Case Material dialog box, 1748
Copy Conditions dialog box, 208, 1805
Copy From dialog box, 1672
core porosity model
default, 780
defining, 780
reading parameters from an external file, 781
Core Porosity Model dialog box, 780, 1676
cores, 1625
Cortex, 49
coupled flow and heat transfer, 695
coupled level set, 1120
coupled solver
Eulerian multiphase model, 1193
coupled solver, pressure-based, 1217
coupled wall interface option, 166, 169, 543
coupled walls, 193, 312
Courant number, 1226
density-based explicit formulation, 1223
density-based implicit formulation, 1223
for VOF calculations, 1143
setting, 1222
Coverage-Dependent Reaction dialog box, 1770
CPD Model dialog box, 1780
Create Surface dialog box, 1409, 1986
Create/Edit Materials dialog box, 805, 885, 1736
Create/Edit Mesh Interfaces dialog box, 169, 524, 541, 1876
creating
conformal periodic boundaries, 191
non-conformal periodic boundaries, 169
crevice model, 936
output file, 940
postprocessing, 938
CRF, 717
critical pressure, 1552
critical specific volume, 1552
critical strain rate, 1552
critical temperature, 1552
Cross model, 416
Cross Model dialog box, 416, 1757
cubic equation of state real gas model
using, 462
cubic equations of state model
Lagrangian dispersed phase model, 465
Postprocessing, 467
curve fitting, 981
Curves dialog box, 1482, 2024
Custom Field Function Calculator dialog box, 1580, 2106
custom field functions, 1552, 1579
exporting, 1579
for solution initialization, 1243
limitations, 1579
sample, 1583
saving, 1582
units of, 125, 1579
Custom Laws dialog box, 2102
Custom Vectors dialog box, 1398, 1973
customizing
field functions, 1579
exporting, 1579
limitations, 1579
GUI, 40
units, 126
vectors, 1398
CutCell Boundary Zones Info dialog box, 621, 1894
CutCell zone remeshing method, 579
CVD, 829
cyclic boundary, 321
cylindrical coordinate system for boundary condition inputs, 259

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylindrical Orthotropic Conductivity dialog box, 426, 1760
cylindrical orthotropic thermal conductivity, 426
Cylindrical Orthotropic UDS Diffusivity dialog box, 431
cylindrical velocities, 1531
  for boundary condition inputs, 266

D

Damkohler number, 1552
Darcy's Law, 222–223, 337
data
  exporting, 84
    after a calculation, 85
    during a transient calculation, 98
    limitations, 84
    steady-state, 97
  importing, 75, 1070
  mapping
    electromagnetic losses, 109
    fluid-structure interaction (FSI) problems, 113
  resetting, 1249
Data Explorer, 84
Data Explorer files, exporting
  during a transient calculation, 98
Data Explorer files, exporting, 84, 90
Data File Quantities dialog box, 121, 1945
data files
  automatic saving of, 65, 548, 627, 1259
  FLUENT UNS, 70
  format, 2187
  options, 121
  parallel, 68
  particle
    importing, 1070
  particle history
    exporting, 97–98
  postprocessing
    time-sequence data, 1500
    with alternative applications, 121
  quantities saved in, 121
  RAMPANT, 70
  reading, 63–65
  writing, 63–65, 183
database, materials, 386, 1057
  data sources, 389
  user-defined, 391
Deactivate Cell Zones dialog box, 196, 2083
deactivating
  cells, 196
  zones, 196
    parallel, 196
decimal separator
  period vs. comma, 1
Decoupled Detailed Chemistry dialog box, 985, 1722
decoupled detailed chemistry model, 985
decoupled flow and heat transfer, 694
Define Event dialog box, 605, 1887
Define Macro dialog box, 1290, 1951
Define Unit dialog box, 127, 1638
Define/Boundary Conditions..., 206, 488, 496
define/boundary-conditions/non-reflecting-bc, 350
Define/Cell Zone Conditions..., 206
Define/Custom Field Functions..., 1580
Define/DTRM Rays... menu, 707
define/dynamic-mesh/actions/remesh-cell-zone-cutcell, 581
define/dynamic-mesh/controls/remeshing-parameter/remeshing-after-moving?, 571
define/dynamic-mesh/controls/spring-on-all-shapes?, 555
Define/General..., 501
Define/Injections... menu, 1038
Define/Materials..., 427, 488, 805
Define/Materials... menu, 387
Define/Mesh Interfaces... menu, 2088
Define/Mixing Planes..., 528
define/mixing-planes/set/conserve-swirl, 532
define/mixing-planes/set/fix-pressure-level, 531
Define/Models..., 506, 510
define/models/acoustics/auto-prune, 1002
define/models/acoustics/cylindrical-export?, 993
define/models/acoustics/export-volumetric-sources?, 993
define/models/acoustics/write-centroid-info, 993
define/models/dpm/options/track-in-absolute-frame, 522, 1008
Define/Models/Energy..., 691
Define/Models/Heat Exchanger/Dual Cell Model..., 787
define/models/heat-exchanger/macro-model/heat-exchanger-macro-report, 798
Define/Models/Multiphase..., 1093
define/models/multiphase/wet-steam/compile-user-defined-wetsteam-functions, 1178, 1181
define/models/multiphase/wet-steam/load-unload-user-defined-wetsteam-library, 1178
define/models/multiphase/wet-steam/set/max-liquid-mass-fraction, 1195
Define/Models/Radiation..., 705, 709, 721, 733
define/models/radiation/s2s-parameters/split-angle, 713
define/models/radiation/solar-parameters/autoread-solar-data/, 759
define/models/radiation/solar-parameters/autosave-solar-data/, 759

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define/Models/Solidification and Melting... menu, 1201
Define/Models/Species..., 895
define/models/species/ecfm-controls, 900
define/models/species/inlet-diffusion?, 884
define/models/viscous/turbulence-expert/turb-non-newtonian?, 678
Define/Operating Conditions..., 405–406, 450, 506, 508
Define/Operating Conditions... menu, 2087
Define/Periodic Conditions..., 494
Define/Phases..., 1098
Define/Profiles..., 368
Define/Turbo Topology..., 1484
Define/Units..., 126
Define/User-Defined/1D Coupling..., 381–382
Define/User-Defined/1D Coupling... menu, 380
define/user-defined/compiled-functions, 478
Define/User-Defined/Execute on Demand... menu, 2113
Define/User-Defined/Fan Model... menu, 360
Define/User-Defined/Function Hooks... menu, 2110
Define/User-Defined/Functions/Compiled... menu, 2109
Define/User-Defined/Functions/Interpreted... menu, 2108
Define/User-Defined/Functions/Manage... menu, 2109
Define/User-Defined/Memory... menu, 2114
Define/User-Defined/Scalars..., 488, 492
Define/User-Defined/Scalars... menu, 2113
definitions
  for field functions, 1529
  for flow variables, 1529
definitions, automatic export
  for particle history data, 102
  for solution data, 100
deformation regions, 631, 634
deforming zones, 613, 619
degrees of freedom, 448
delayed detached eddy simulation (DDES), 674
Delete Cell Zones dialog box, 196, 2082
deleting
  objects from a scene, 1452
  zones, 196
Delta-Eddington Scattering Function dialog box, 1777
Delta-Eddington scattering phase function, 438
dense discrete phase model
  postprocessing, 1199
density, 403, 818, 1553
  Boussinesq model, 403
  composition-dependent, 407
  constant, 403–404
  discrete phase, 1059
  ideal gas law, 403, 406
  incompressible ideal gas law, 403, 405
  interpolation schemes, 1213
  parameterized, 403
  restrictions, 403
  temperature-dependent, 403–404
density-based solver, 286, 1207
  Courant number, 1222
  explicit
    Courant number, 1223
  explicit relaxation, 1225
  flux types, 1224
  implicit
    Courant number, 1223
  limitations, 1207
  residuals, 1270
derivatives, 1553
DES turbulence model, 650
Detached Eddy Simulation (DES) turbulence model, 650
detailed chemistry model, decoupled, 985
devolatilization, 1554
  models, 1061
dialog box, 27
dialog box layout, 1432
diffusion, 142, 427
  species, 694
  thermal, 1574
diffusion coefficient, 485, 800, 818
  species, 1555, 1559
  thermal, 442
  user-defined scalar, 1553
diffusion-based smoothing method, 556
diffusivity, binary, 1060
dimension reduction, 924, 927
direct search methods, 632
disabling graphics windows, 1410
discrete ordinates (DO) radiation model
  angular discretization, 720
  boundary conditions
    opaque walls, 726
    semi-transparent walls, 728
    walls, 725–726
  energy coupling
    enabling, 723
  non-gray
    absorption coefficient, 724
    black body emission factor, 722
    emissivity weighting factor, 722
    inputs for, 721
    refractive index, 724
  properties, 435
  residuals, 735
  setting up, 720
  solution parameters, 735
discrete phase, 1007

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

aborted trajectories, 1012
absorption coefficient, 1553
accretion, 1017, 1054, 1553
  reporting, 1087
air-blast atomizer
  injections, 1032
boundary conditions, 1050
  escape, 1051
  interior, 1052
  reflect, 1050
  trap, 1051
  wall jet, 1052
  wall-film, 1052
Brownian force, 1016
burnout, 1553
cloud tracking, 1045
coal combustion, 858
coefficient of restitution, 1050, 1053
combusting, 1025, 1057
concentration, 1081, 1553
cone injections, 1024, 1027
coupled calculations, 1065, 1067
coupled heat-mass solution, 1021
customized particle laws, 1042, 1046
devolatilization, 1554
display, 1068
drag coefficient, 1014
droplet, 1025, 1057
effervescent atomizer
  injections, 1034
emission, 1553
erosion, 1017, 1054, 1554
  reporting, 1087
evaporation, 1554
file
  injections, 1034
flat-fan atomizer
  injections, 1033
group injections, 1024, 1027
heat transfer, 1009, 1073
in multiple reference frames, 522, 1008
inert, 1025, 1057
initial conditions, 1022
  file, 1024
  read from a file, 1025
injections, 1024
  copying, 1039
  creating, 1039
  deleting, 1039
  file, 1024
  listing, 1040
  read from a file, 1025
  reading, 1040
  setting properties, 1040
  setting properties for multiple, 1047
  writing, 1040
inputs, 1009
  for boundary conditions, 1050
  for initial conditions, 1022
  for material properties, 1054
  for optional models, 1015
  for particle tracking, 1010
  for spray modeling, 1017
  for transient particles, 1010
interaction with continuous phase, 1010
lift force, 1017
limitations of, 522, 1008
mass flow rate, 1026, 1028, 1037
mass transfer, 1009, 1051, 1060, 1073
materials, 387, 1042, 1054, 1057
model compatibility, 2183
multicomponent, 1025, 1057
numerics, 1019
options, 1015
overview, 1008–1009
parallel processing, 1087
  hybrid, 1087
  message passing, 1087
  shared memory, 1087
  workpile, 1087
particle cloud tracking, 1045
particle stream, 1040
particle summary reporting, 1085
particle tracking, 1009
plain-orifice atomizer
  injections, 1030
point properties, 1026
postprocessing, 1067
pressure-swirl atomizer
  injections, 1031
properties, 1054
radiation heat transfer to, 1016, 1063
reference frame, 1021
residence time, 1074
restitution coefficient, 1050, 1053
scattering, 1554
setup, 1038
size distribution, 1035
solution procedures, 1063
sources, 1554
staggering, 1022
stochastic tracking, 1044, 1066
summary report, 1072
surface injections, 1024, 1029

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

thermophoretic coefficient, 1059
thermophoretic force, 1016
time step, 1012
tracking schemes, 1020
tracking, parameter, 1012
trajectory calculations, 1064
trajectory reporting, 1072, 1078
at boundaries, 1082
unsteady tracking, 1080
transient particles, 1010
turbulence, 1017
uncoupled calculations, 1064
under-relaxation, 1066
user-defined functions, 1018, 1046
using, 1009
vaporization, 1051
wet combustion, 1042
Discrete Phase dialog box, 1170, 1789
Discrete Phase Model dialog box, 1012, 1015, 1038, 1069, 1723
discrete random walk (DRW) model, 1044
discrete transfer radiation model (DTRM)
clusters
controlling, 708
displaying, 709
properties, 435
rays
controlling, 708
defining, 707
reading and writing files, 709
residuals, 735
setting up the model, 707
solution parameters, 733
discretization, 142
first-order scheme, 1211
first-to-higher order blending, 1211
inputs for, 1210
power-law scheme, 1212
QUICK scheme, 1212
second-order scheme, 1211
Display Options dialog box, 1412, 1414, 1423, 1425, 1992
display properties, 1446
Display Properties dialog box, 1446, 1996
Display/Annotate..., 1415
Display/Annotate... menu, 1415
Display/Colormap..., 1417
Display/Colormap... menu, 1417
Display/Contours..., 1387
Display/Contours... menu, 1386
Display/DTRM Graphics... menu, 738
Display/Import Particle Data... menu, 2144
Display/Lights..., 1423
Display/Lights... menu, 1423
Display/Mesh..., 1381
Display/Mesh... menu, 1379
Display/Mouse Buttons..., 1428
Display/Mouse Buttons... menu, 1427
Display/Options..., 1412, 1414, 1423, 1425
Display/Options... menu, 1411
Display/Pathlines..., 1401
Display/Pathlines... menu, 1400
Display/PDF Tables/Curves..., 874
Display/Residuals..., 1272
Display/Residuals... menu, 1478
Display/Scene Animation... menu, 1454
Display/Scene..., 1411, 1444
Display/Scene... menu, 1444
display/set/title, 1414
display/set/windows/scale, 1414
display/set/windows/text, 1414
display/set/zero-angle-dir, 1484
Display/Sweep Surface..., 1409
Display/Sweep Surface... menu, 1409
Display/Vectors..., 1394
Display/Vectors... menu, 1393
Display/Video Control... menu, 1458
Display/Views..., 1434, 1439, 1441
Display/Views... menu, 1433
Display/Zone Motion..., 626
Display/Zone Motion... menu, 626
displaying
animations, 1454
restrictions, 1458
annotations, 1415
bounding frames, 1452
captions, 1413
colormap, 1417
colors, 1447
contours, 1386
for turbomachinery, 1496–1497
creating surfaces for, 1355
graphics, 1379
in multiple graphics windows, 1412
legends, 1413
lights, 1423
meshes, 1379
automatically, 2, 36
mirrored domains, 1441
on surfaces with uniform distribution, 1361, 1364
on sweep surfaces, 1409
outlines, 1379, 1384
overlaid graphics, 1411
pathlines, 1400

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

periodic repeats, 1441
profiles, 1386
properties, 1446
rendering options, 1425
scenes, 1444
symmetry, 1441
titles, 1414, 1473
transparency, 1447
turbomachinery
2D contours, 1497
averaged contours, 1496
vectors, 1393
views, 1433
visibility, 1446
dissipation, 1238
divergence, 1217
troubleshooting, 1235
domain, reordering, 199
donor-acceptor scheme, 1095
double buffering, 1425
double-precision solvers, 1, 1269
dough, 418
drag coefficient, 1278, 1515
discrete phase, 1014
Drag Monitor dialog box, 1279, 1918
drop-down list, 30
droplet, 1007, 1025, 1057
initial conditions, 1022
size distribution, 1035
trajectories
alphanumeric reporting, 1081
display of, 1068
DRW model, 1044
DTRM Graphics dialog box, 738, 2142
DTRM Rays dialog box, 707, 2103
dual cell
restrictions, 771
Dual Cell Heat Exchanger dialog box, 787, 1667
duals, 178
duplicate shadow nodes, 174
dynamic head, 330
dynamic layering method, 564
Dynamic Mesh Events dialog box, 605, 1886
dynamic mesh model
model compatibility, 2183
partitioning in parallel, 1610, 1613
Dynamic Mesh task page, 550, 1877
Dynamic Mesh Zones dialog box, 613, 1889
dynamic meshes
2.5D model, 583
boundary layer smoothing, 561
constraints, 555, 567, 571, 578, 580, 583, 586
crevice model, 936
defining events, 605
diffusion-based smoothing method, 556
dynamic layering, 564
dynamic layering method, 564
IC3M files, 79
implicit update, 550
settings, 603
in-cylinder motion, 591
in-cylinder option, 550
settings, 587
Laplacian smoothing, 551, 560
mesh motion methods, 551
feature detection, 586
volume mesh update, 586
mesh requirements
2.5D surface remeshing, 583
cell zone remeshing, 571
CutCell zone remeshing, 580
dynamic layering, 567
face region remeshing, 578
feature detection, 586
local face remeshing, 571
spring-based smoothing, 555
previewing, 626
remeshing, 567
remeshing methods, 567
2.5D surface, 582
cell zone, 571
CutCell zone, 579
face region, 577
local, 570
local cell, 570
local face, 571
setting parameters, 550
setup, 548
Six DOF solver, 550, 602
settings, 601
smoothing, 551
solid-body kinematics, 623
specifying zone motion, 613
spring-based smoothing, 551
spring-based smoothing method, 552, 555
steady-state, 628
additional local remeshing, 571
dynamic pressure, 1555

E

EDC model
model compatibility, 2183
eddy-dissipation model
model compatibility, 2183

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit Automatic Initialization and Case Modifications dialog box, 1295
Edit Material dialog box, 1782
Edit Property Methods dialog box, 1749
effective Prandtl number, 1555
effective thermal conductivity, 1555
effective viscosity, 1555
effectiveness, 779
efficiencies
  for pumps and compressors, 1491
  for turbines, 1494
elastic collision, 1051
elasticity modulus, 1148, 1158, 1788, 1791
electromagnetic losses, mapping, 109
Embed graphics window, 1432
embed graphics window menu, 1431
Embedded LES (E-LES) turbulence model, 651
embedding graphics windows, 2
emissivity, 724–725
emissivity weighting factor, 706, 722
empirical fuel, 846, 854
energy, 1559, 1574
  parameter, 448
  sources, 247
Energy dialog box, 506, 691, 698, 1645
energy source terms, defining, 247
energy sources
  parameterized, 247
engine ignition
  auto, 933
  spark, 929
EnSight Case Gold files, exporting
  during a transient calculation, 98
EnSight Case Gold files, exporting, 84, 91
EnSight files
  importing, 78
  pathlines, 1407
EnSight particle files, exporting
  during a transient calculation, 98
  from a steady-state solution, 97
enthalpy, 1555, 1569
  standard-state, 819
entropy, 819, 1523, 1556
environment variables, 8
equation of state, 456
Equations dialog box, 1318, 1904
equilibrium chemistry, 839, 843
  condensed phase species, 854
  empirical fuels, 846
Ergun equation, 232
erosion, 1017, 1054, 1554
  reporting, 1087
Error dialog box, 32
error messages, 32
error reduction, 1211
Eulerian model
  time-dependent calculations
    inputs for, 1163
Eulerian multiphase flow, model compatibility, 2183
Eulerian multiphase model
  drag function, 1160
  lift forces, 1161
  restitution coefficients, 1161
  solution method
    coupled solver, 1193
  solution strategies, 1154, 1192
  virtual mass force, 1163
evaporation, 1554
Events Preview dialog box, 605, 1889
events, defining dynamic mesh, 605
Execute Commands dialog box, 1288, 1950
Execute on Demand dialog box, 2113
exhaust fan boundary, 249, 299
Exhaust Fan dialog box, 299, 1815
existing value, 1557
exiting, 19
explicit interpolation scheme, 1095
explicit relaxation
  density-based solver, 1225
explicit time stepping
  restrictions, 1225
exploded views, 1449
Export dialog box, 85, 2056
Export Particle History Data dialog box, 97, 2060
Export to CFD-Post dialog box, 2061
exporting
  ABAQUS files, 84, 87, 98
  ANSYS CFD-Post, 104
  ANSYS CFD-Post compatible files, 84, 89
  ASCII files, 84, 89, 98
  AVS files, 84, 89, 98
  CGNS files, 84, 90, 98
  Data Explorer files, 84, 90, 98
  EnSight Case Gold file, 84, 91, 98
  EnSight particle files
    during a calculation, 98
    from a steady-state solution, 97
  FAST files, 84, 93, 98
  FAST Solution files, 84, 93, 98
  Fieldview Unstructured files, 84, 94, 98
  files, 84
    after a calculation, 85
    during a transient calculation, 98
    limitations, 84

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

force data, 84
heat flux data, 696
I-deas Universal files, 84, 94, 98
Mechanical APDL files, 84, 87
Mechanical APDL Input files, 84, 88, 98
NASTRAN files, 84, 95, 98
particle history data
steady-state, 97
transient, 98
PATRAN files, 84, 95, 98
polyhedral meshes, 84
RadTherm files, 84, 96, 98
Tecplot files, 84, 96, 98
extended coherent flame model
using, 899
external flows, 290
boundary conditions, 255
external heat transfer boundary conditions, 318
extruding face zones, 194

F

face area, 1579
face area magnitude, 1557
face handedness, 172, 1557
repairing, 174
face node order, 172
repairing, 174
face region remeshing method, 577
face swapping, 1353
face zones
extruding, 194
fusing, 190
orienting, 194
separating, 185–186
slitting, 193
face-node connectivity, 136
false diffusion, 142
Fan dialog box, 325, 1819
fan model, 194, 324
user-defined, 324, 359
fans
postprocessing for, 1392, 1409, 1472
far-field boundary, 249, 268, 290
FAST, 84
FAST files, exporting
during a transient calculation, 98
Fast Fourier Transform (FFT)
customizing the input, 1504
customizing the output, 1505
limitations, 1501
loading data, 1504
postprocessing, 1500
using, 1503
windowing, 1501
FAST Solution files, exporting
during a transient calculation, 98
FAST Solution files, exporting, 84, 93
FAST files, exporting, 84, 93
fe2ram, 155
feature detection, 586
feature outline, 1384
Fick's law, 439
Field Function Definitions dialog box, 1582, 2107
field functions
custom, 1579
exporting, 1579
limitations, 1579
sample, 1583
saving, 1582
units of, 125, 1579
definitions, 1529
for solution initialization, 1243
Fieldview, 84
Fieldview Unstructured files, exporting
during a transient calculation, 98
Fieldview Unstructured files, exporting, 84, 94
FIELDVIEW files, pathlines saved in, 1406
File XY Plot dialog box, 1472, 2016
File/Batch Options... menu, 16
file/confirm-overwrite?, 61
File/Data File Quantities... menu, 121
File/Exit menu, 19
File/Export to CFD-Post... menu, 104
file/export/custom-heat-flux, 696
file/export/mechanical-apdl, 87
File/Export/Particle History Data... menu, 97
File/Export/Solution Data..., 85
File/Export/Solution Data... menu, 85
File/FSI Mapping/Surface..., 113
File/FSI Mapping/Surface... menu, 113
File/FSI Mapping/Volume..., 113
File/FSI Mapping/Volume... menu, 113
File/Import/ABAQUS/Filbin File... menu, 77
File/Import/ABAQUS/Input File...,
File/Import/ABAQUS/Input File...File/Import/ABAQUS/Input File... menu, 77
File/Import/ABAQUS/ODB File...,
File/Import/ABAQUS/ODB File...File/Import/ABAQUS/Filbin File...,
File/Import/ABAQUS/ODB File...File/Import/ABAQUS/Filbin File...File/Import/ABAQUS/ODB File... menu, 77
File/Import/CFX, 154
File/Import/CFX/Result File...,
File/Import/CFX/Result File... menu, 77

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File/Import/CFX/Result File...File/Import/CFX/Definition File...,
File/Import/CFX/Result File...File/Import/CFX/Definition File...File/Import/CFX/Definition File... menu, 77
File/Import/CGNS/Data...,
File/Import/CGNS/Data... menu, 78
File/Import/CGNS/Data...File/Import/CGNS/Mesh and Data...,
File/Import/CGNS/Data...File/Import/CGNS/Mesh and Data...File/Import/CGNS/Mesh...,
File/Import/CGNS/Data...File/Import/CGNS/Mesh and Data...File/Import/CGNS/Mesh...File/Import/CGNS/Mesh... menu, 78
File/Import/CGNS/Mesh and Data... menu, 78
File/Import/CHEMKIN Mechanism..., 84, 827, 832
File/Import/CHEMKIN Mechanism... menu, 84
File/Import/EnSight..., 78
File/Import/EnSight... menu, 78
File/Import/FIDAP..., 79, 157
File/Import/FIDAP... menu, 79
File/Import/FLUENT 4 Case File..., 156
File/Import/FLUENT 4 Case File... menu, 83
File/Import/GAMBIT..., 79
File/Import/GAMBIT... menu, 79
File/Import/HYPERMESH ASCII..., 79
File/Import/HYPERMESH ASCII... menu, 79
File/Import/I-deas Universal..., 150
File/Import/I-deas Universal...File/Import/I-deas Universal... menu, 80
File/Import/IC3M..., 79
File/Import/IC3M... menu, 79
File/Import/LSTC/State File..., 80
File/Import/LSTC/State File... menu, 80
File/Import/Marc POST..., 81
File/Import/Marc POST... menu, 81
File/Import/Mechanical APDL/Input File..., 81
File/Import/Mechanical APDL/Input File... menu, 81
File/Import/Mechanical APDL/Result File...,
File/Import/Mechanical APDL/Result File...File/Import/Mechanical APDL/Result File... menu, 81
File/Import/NASTRAN, 151
File/Import/NASTRAN/Bulkdata File..., 81
File/Import/NASTRAN/Bulkdata File... menu, 81
File/Import/NASTRAN/Op2 File..., 81
File/Import/NASTRAN/Op2 File... menu, 81
File/Import/Partition/Metis Zone... menu, 1622
File/Import/Partition/Metis... menu, 1622
File/Import/PATRAN, 152
File/Import/PATRAN/Neutral File..., 82
File/Import/PATRAN/Neutral File... menu, 82
File/Import/PATRAN/Result File..., 82
File/Import/PATRAN/Result File... menu, 82
File/Import/PLOT3D Grid..., 82
File/Import/PLOT3D/Grid File... menu, 82
File/Import/PreBFC File..., 83, 149
File/Import/PreBFC File... menu, 83
File/Import/PTC Mechanica Design..., 82
File/Import/PTC Mechanica Design... menu, 82
File/Import/Tecplot..., 83
File/Import/Tecplot... menu, 83
File/Interpolate... menu, 106
file/read-macros, 1291
file/read-settings, 72
file/read-transient-table, 378, 1049
File/Read/Case and Data..., 65
File/Read/Case and Data... menu, 63
File/Read/Case... menu, 63
File/Read/Case...File/Write/Case..., 64
File/Read/Data..., 64, 69
File/Read/Data... menu, 63
File/Read/DTRM Rays... menu, 709
File/Read/Journal...,
File/Read/Journal... menu, 73
File/Read/Mesh..., 62–63
File/Read/PDF... menu, 886
File/Read/Profile..., 70, 368
File/Read/Profile... menu, 70
File/Read/Scheme... menu, 73
File/Read/View Factors... menu, 720
File/RSF... menu, 13
File/Save Picture..., 118
File/Save Picture... menu, 117
file/set-batch-options, 16
file/write-macros, 1291
file/write-settings, 72
file/write-surface-clusters/split-angle, 713
File/Write/Autosave..., 548
File/Write/Boundary Mesh... menu, 72
File/Write/Case and Data..., 65
File/Write/Case and Data... menu, 63
File/Write/Case...,
File/Write/Case... menu, 63
File/Write/Data..., 64, 68
File/Write/Data... menu, 63
File/Write/Flamelet..., 867
File/Write/Flamelet... menu, 2050
File/Write/PDF..., 874
File/Write/PDF... menu, 2050
File/Write/Profile..., 71
File/Write/Profile... menu, 70
File/Write/Start Journal... menu, 73
File/Write/Start Journal...File/Write/Stop Journal-File/Read/Journal..., 74
File/Write/Stop Journal menu, 73

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File/Write/Stop JournalFile/Read/Journal...,
File/Write/Stop Transcript,
File/Write/Stop Transcript menu, 75
File/Write/Stop TranscriptFile/Write/Start Transcript...,
File/Write/Stop TranscriptFile/Write/Start Transcript...File/Write/Start Transcript... menu, 75
File/Write/Surface Clusters..., 718
File/Write/Surface Clusters... menu, 711, 2050
filenames , 51
files, 57
  ABAQUS
    exporting, 84, 87, 98
    importing, 77
    mapping data with, 113
  ANSYS CFD-Post compatible, exporting, 84, 89
  ANSYS CFX
    importing, 77
  ANSYS FIDAP, importing, 79
  ASCII, exporting, 84, 89, 98
  asynchronous, 63, 69
  automatic exporting of, 98
  automatic saving of, 65, 548, 627, 1259
  AVS, exporting, 84, 89, 98
  binary, 59, 63
  boundary mesh, 72
  case, 63–64
    autosaving, 65
  CGNS
    exporting, 84, 90, 98
    importing, 78
  chemical mechanism, 84, 827, 848
  compressed
    reading, 59
    writing, 60
  data, 63–64
    autosaving, 65
    options, 121
    parallel, 68
    postprocessing with alternative applications, 121
    quantities saved in, 121
  Data Explorer, exporting, 84, 90, 98
  DTRM rays, 709
  EnSight, 1407
    importing, 78
  EnSight Case Gold, exporting, 84, 91, 98
  EnSight particle, exporting
    during a calculation, 98
    from a steady-state solution, 97
  exporting, 84
    after a calculation, 85
    during a transient calculation, 98
    for steady-state calculations, 97
    limitations, 84
    polyhedral meshes, 84
  FAST Solution, exporting, 84, 93, 98
  FAST, exporting, 84, 93, 98
  Fieldview Unstructured, exporting, 84, 94, 98
  FIELDVIEW, pathlines saved in, 1406
  flamelet, 879
  fluid-structure interaction (FSI), 113
  format, 2187
    detecting, 59
  formatted, 59, 63
  GAMBIT
    importing, 79
    meshes, 148
    pathlines, 1407
    reading, 63
  GeoMesh
    importing, 79
    meshes, 149
    reading, 63
  HYPERMESH ASCII, importing, 79
  I-deas Universal
    exporting, 84, 94, 98
    importing, 80
    mapping data with, 113
  IC3M, importing, 79
  importing, 75
    particle data, 1070
  interpolation, 108
  journal, 73
  log, 73
  loss, 112
  LSTC, importing, 80
  Marc POST, importing, 81
  Mechanical APDL
    exporting, 84, 87
    importing, 81
    mapping data with, 113
  Mechanical APDL Input, exporting, 84, 88, 98
  mesh, 62
  MPEG, 1303, 1457
  naming
    conventions, 58
    options, 61
  NASTRAN
    exporting, 84, 95, 98
    mapping data with, 113
  NASTRAN, importing, 81
  numbering, 61
  overwriting, 61
  parallel data, 68
  pathline, 1405

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EnSight format, 1407
FIELDVIEW format, 1406
GAMBIT format, 1407
PATRAN
exporting, 84, 95, 98
importing, 82
mapping data with, 113
picture, 117
PLOT3D, importing, 82
PreBFC meshes, 149
unstructured, 63
profile, 70
PTC Mechanica Design
importing, 82
study, 83
RadTherm, exporting, 84, 96, 98
ray, 709
ray tracing, 707
reading, 57
compressed, 59
shortcuts, 57
toolbar buttons, 61
reading multiple, 36
recently read, 59
searching for, 35
selecting, 33
shortcuts for reading and writing, 57
suffixes, 58
surface mechanism, 832
surface mesh, 72
Tecplot
importing, 83
Tecplot, exporting, 84, 96, 98
text, 59, 63
TGrid meshes, 149
transcript, 75
transfer, 109
unformatted, 59, 63
writing, 57
compressed, 59
shortcuts, 57
toolbar buttons, 61
XML, 1598
filled
contours, 1390
meshes, 1383, 1446
profiles, 1390
filter papers, 221, 234
filters
fe2ram, 155
fl42seg, 149, 156
partition, 1622
tmerge, 159
tpoly, 155
fine scale
mass fraction, 1557
temperature, 1557
transfer rate, 1557
finite-rate reactions
particle surface, 835
volumetric, 799
wall surface, 829
first-order accuracy, 1211
first-to-higher order blending, 1211
fixing variable values in cell zones, 241
fl42seg, 149, 156
flame speed model
Peters, 898
Zimont, 898
Flamelet 2D Curves dialog box, 1699
Flamelet 3D Surfaces dialog box, 868, 1697
flamelet model
files, 879
standard, 879
flamelet generation approach
inputs for, 847
importing a flamelet file, 849
inputs for, 839
look-up tables, 874
parameters, 863
PDF table parameters, 871
setting up, 847
standard format files, 879
steady laminar
automated grid refinement, 865
setting up, 847
unsteady laminar
diesel, 850
setting up, 847
using, 850
zeroing species, 864
flexible-cycle multigrid, 1226
floating operating pressure, 507
flow direction
at mass flow inlets, 274
Cartesian, 274
cylindrical, 274
local cylindrical, 274
local cylindrical swirl, 274
at pressure far-field boundaries, 292
at pressure inlets, 259
Cartesian, 259
cylindrical, 259
local cylindrical, 259

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

local cylindrical swirl, 259
flow distributors, 221
flow rate
for multiphase calculations, 1200
flow time
appending to file name, 65, 100
flow variable definitions, 1529
fluent -help, 10
FLUENT 4 case files, 83, 156
Fluent Database Material dialog box, 1783
Fluent Database Materials dialog box, 389, 807
FLUENT Database Materials dialog box, 1744
FLUENT Launcher, 2, 1432
environment options, 8
general options, 3
parallel options, 5, 1587
remote options, 6, 1591
scheduler options, 7, 1589
UDF options, 8
FLUENT UNS case and data files, 70
FLUENT/UNS case files, 156
Fluid dialog box, 215, 225, 488, 1796
fluid flow
compressible, 503
inviscid, 510
periodic, 493
swirling and rotating, 497
fluid materials, 216, 387, 807
fluid zone, 214
fluid-structure interaction (FSI) simulations, 113, 603
flux
reports, 1510
through boundaries, 1510
types
AUSM, 1224
low diffusion Roe-FDS, 1224
Roe-FDS, 1224
Flux Reports dialog box, 1511, 2027
FMG multigrid, 1244
convergence strategies, 1245
using, 1244
force report, 1515
Force Reports dialog box, 511, 1515, 2029
forces
coefficients of, 1515, 1523
monitoring, 1278
exporting data, 84
on boundaries, 1515
formation enthalpy, 447, 819
formation rate, 1564
formats, file
case and data files, 2187
exporting, 84, 98
importing, 75
Fourier Transform dialog box, 1503, 2019
Fractional Step algorithm, 1216
free stream, 290
friction packing limit, 1148, 1158
frictional modulus, 1148, 1157
frictional pressure, 1148, 1157
frictional viscosity, 1157
frozen flux formulation, 1259
fuel NOx parameters, 949
fuel streams
NOx model, 946
limitations, 947
SOx model, 961
limitations, 962
full multicomponent diffusion, 440
full multigrid (FMG), 1244
convergence strategies, 1245
using, 1244
fully-developed flow, 493
Fuse Face Zones dialog box, 190, 2080
fusing face zones, 190
FW-H acoustics model, 989

G

G-equation, 896
GAMBIT files
importing, 79
meshes, 148
pathlines, 1407
reading, 63
gas constant, 1557
gas law, 403, 405, 407, 449, 506
gaseous and liquid fuel NOx parameters, 950
gauge pressure, 449
general non-reflecting boundary conditions
overview of, 351
theory, 352
using, 357
General task page, 501, 1209, 1631
GeoMesh mesh files, 149
importing, 79
reading, 63
geometric reconstruction scheme, 1095
Geometry Based Adaption Controls dialog box, 2133
Geometry Based Adaption dialog box, 1341, 2132
geometry-based adaption, 160, 1341
global matrix size, 1624
glossary, 2205
gnuplot, 940
governing equations

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

source term additions to, 244
gradient adaption, 1331
dynamic, 1333
Gradient Adaption dialog box, 1331, 2119
gradient option
setting, 1209
granular bulk viscosity, 1157
granular conductivity, 1158, 1557
granular temperature, 1148, 1158
granular viscosity, 1148, 1157
graphical user interface (GUI), 21
customizing the, 40
graphics, 1379
overlaying, 1411
picture files, 117
window dumps, 121
Graphics and Animations task page, 1965
graphics device information, 1426
Graphics Periodicity dialog box, 1443, 2003
graphics window menu, 1431
graphics windows, 36
active, 1413
captions, 1413
closing, 1412
color scheme, 2
embedding, 2
hiding, 1410
multiple, 1412
opening, 1412
title, 1414
Windows features, 37
gravitational acceleration, 698
Gray-Band Absorption Coefficient dialog box, 1776
Gray-Band Refractive Index dialog box, 1777
grid
case check, 1306
coarse levels, 1225
grid resolution RANS turbulence models, 647
Grid Resolution SRS turbulence models, 651
free shear flows, 651
wall boundary layers, 651
GT-Power, 379
GUI, 21
customizing the, 40

H

hanging nodes, 129
effect on polyhedral conversion, 181–182
removing, 155
HCN density, 1558
heat capacity, 433
heat exchanger groups, 769
Heat Exchanger Model dialog box, 771, 781, 1666
heat exchanger models, 330, 769
dual cell
inputs, 787
restrictions, 771
effectiveness, 779
inputs, 771
macros, 777
ntu-model, 769
features, 769
restrictions, 770
reports, 794
restrictions, 770–771
simple-effectiveness-model, 769
features, 769
restrictions, 770
Heat Exchanger Report dialog box, 2040
heat flux, 319, 1510, 1575
exporting data, 696
wall, 317
heat of formation, 447
heat of heterogeneous reaction, 1558
heat of reaction, 815, 1558
surface reaction, 831
heat sources, 219, 244, 247, 330
heat transfer, 691
boundary conditions, 318
at walls, 309
buoyancy-driven flows, 697
convective and conductive, 691
inputs for, 691
modeling, 691
natural convection flows, 697
overview of models, 691
periodic, 763
postprocessing, 695
radiation, 318, 704
rate, 1511
reporting, 695
averaged coefficients, 696
for radiation, 736
through boundaries, 696
through surface, 696
shell conduction considerations, 701
solution
procedures, 691, 693
strategies, 693
to the wall, 317
volume sources, 219, 244
heat transfer coefficient, 146, 1523, 1573
calculations of, 319
Heat Transfer Data Table dialog box, 772, 1670

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

helicity, 1558
help, 41
  text user interface, 55
help browser, 41
help viewer, 41
Help/Context-Sensitive Help menu, 42
Help/License Usage... menu, 44
Help/More Documentation... menu, 44
Help/Online Technical Support... menu, 44
Help/User's Guide Contents... menu, 42
Help/User's Guide Index... menu, 43
Help/Using Help... menu, 44
Help/Version... menu, 45
Herschel-Bulkley dialog box, 418, 1758
Herschel-Bulkley model, 417
heterogeneous reactions
  defining, 1101
hex cells, 140
hexcore meshes
  limitations, 181–182
  removing hanging nodes, 155
HFSS Mapping dialog box, 2065
hidden line removal, 1425
hidden surface removal, 1425
hiding graphics windows, 1410
Histogram dialog box, 1478, 1522, 2015
histogram plots
  options, 1479
histograms, 1522
  plots, 1467, 1478
  reports, 1522
Hosts Database dialog box, 2166
hybrid initialization, 1245
  solution strategies, 1247
  using, 1245
Hybrid Initialization dialog box , 1941
hybrid RANS-LES turbulence models, 649
hydraulic diameter, 251
HYPERMESH ASCII files, importing, 79

I

I-deas, 84
I-deas Universal files
  exporting, 84, 94
    during a transient calculation, 98
  importing, 80
  mapping data with, 113
I-deas Universal files, 150
IC3M files, importing, 79
ICEM CFD mesh file, 150
ideal gas law, 290, 405, 407, 449, 506, 1025
ignition source, 824
impeller-ba e interaction, 522
implicit interpolation scheme, 1097
implicit update option, 550
  settings, 603
Import Particle Data dialog box, 2144
importing
  ABAQUS files, 77
  ANSYS CFX files, 77
  ANSYS FIDAP files, 79
  CGNS files, 78
  CHEMKIN files, 84, 985
  EnSight files, 78
  fe2ram filter, 155
  files, 75
  GAMBIT files, 79
  GeoMesh files, 79
  HYPERMESH ASCII files, 79
  I-deas Universal files, 80
  IC3M files, 79
  LSTC files, 80
  Marc POST files, 81
  Mechanical APDL files, 81
  NASTRAN files, 81
  particle data, 1070
  PATRAN files, 82
  PLOT3D files, 82
  PTC Mechanica Design files, 82
  Tecplot files, 83
improving meshes, 174
in-cylinder model
  crevice model
    output file, 940
    postprocessing, 938
  IC3M files, 79
in-cylinder option, 550, 591
  events, 607
  geometry and motion attributes, 594
  mesh topology, 592
  settings, 587
  valve opening and closing, 601
In-Cylinder Output Controls dialog box, 1884
incident radiation, 1558, 1573
  reporting, 739
incomplete particle trajectories, 1013
incompressible flow calculations at inlet pressure boundaries, 263
Inert dialog box , 1705
inertial resistance, alternative formulation for, 228
information
  graphics device, 1426
Information dialog box, 31
initial conditions, 1239

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

discrete phase, 1022
initializationr
hybrid, 1245
initializing the solution, 1239
using average values, 1241
injections
copying, 1039
creating, 1039
deleting, 1039
listing, 1040
modifying, 1039
reading, 1040
writing, 1040
Injections dialog box, 1038, 2096
inlet boundaries, 264
inlet vent boundary, 249, 278
Inlet Vent dialog box, 278, 1821
Input Parameter Properties dialog box, 2043
Input Summary dialog box, 1525, 2158
input summary report, generating a, 1525
intake fan boundary, 249, 280
Intake Fan dialog box, 280, 1826
integer entry, 29
integral reporting, 1285, 1288, 1519, 1521
integration parameters
setting, 924
Integration Parameters dialog box, 914–915, 1694
interconnects, 1585
Linux/UNIX, 1601
Windows, 1596
Interface dialog box, 1831
interface zone, 169, 537, 541
contour display, 547
interfaces, non-conformal, 161
Interior Cell Zone Selection dialog box, 995, 1734
Interior dialog box, 1831
interior zone, 91, 205, 325, 333
intermittency factor, 1559
internal emissivity, 726, 730
internal energy, 1559
Interpolate Data dialog box, 107, 2063
interpolation, 106
file format, 108
for electromagnetic losses, 109
for fluid-structure interaction (FSI) problems, 113
mesh-to-mesh, 106
Interpreted UDFs dialog box, 2108
interrupt
of calculations, 1249
interrupts , 53
Intrinsic Combustion Model dialog box, 1062, 1781
inviscid flows, 510
inputs for, 510
solution strategies for, 511
ISAT, 924
dimension reduction, 924, 927
Iso-Clip dialog box, 1373, 1935
Iso-Surface dialog box, 1371, 1933
Iso-Value Adaption dialog box, 1335, 2121
Iso-Value dialog box, 1450, 1998
isosurfaces, 1355, 1371
isotropic diffusivity, 485
isotropic scattering phase function, 438
isovalue, 1371, 1373
adaption, 1335
modification, 1450
iteration, 1247
appending to file name, 65
control during, 1288
interrupt of, 1249
iterative procedure, 1247

J

jet acoustic power, 1559
jet acoustic power in dB, 1559
jet breakup, 1097
journal files, 73
jump
adhesion
inputs for, 1117, 1141

K

k-epsilon model
model compatibility, 2183
k-omega model
model compatibility, 2183
k-ε turbulence models, 645
k-ε multiphase turbulent viscosity, customizing, 1165
k-ω turbulence models, 646
key frame, 1454
kinetic energy
subtest, 1572
kinetic reaction rate, 1559
kinetic theory, 412, 420, 434, 444
parameters, 448
Kinetics/Diffusion-Limited Combustion Model dialog box, 1062, 1780

L

labels, 2023
labels, colormap, 1419
Lagrangian discrete phase model, 1007
Lagrangian dispersed phase model

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cubic equations of state model, 465
laminar finite-rate model
model compatibility, 2183
laminar flow shear stress, 316
laminar kinetic energy, 1560
laminar viscosity, 1561
laminar zone, 216, 237
laminar-turbulent transition models, 646
Laplacian smoothing, 560, 1351
large eddy simulation (LES)
model compatibility, 2183
reporting mean flow quantities, 689
solution strategies, 684
Large Eddy Simulation (LES) turbulence model, 648
latent heat, 1201
of vaporization, 858, 1059
layout, dialog box and window, 1432
LEE self-noise source, 1560
LEE shear-noise source, 1560
LEE total noise source, 1560
legend, 1413, 1473
length scale, 251
Lennard-Jones parameters, 412, 444, 448
LES
model compatibility, 2183
LES turbulence model, 648
Lewis number, 831
license users, 44
lift coefficient, 1278, 1515
lift force, 1017
Lift Monitor dialog box, 1279, 1920
lights, 1423, 1446
Lights dialog box, 1423, 2005
Lilley's self-noise source, 1560
Lilley's shear-noise source, 1560
Lilley's total noise source, 1560
limiter, 1239
line surfaces, 1355, 1361
using the line tool, 1363
line tool, 1362–1363
Line/Rake Surface dialog box, 1361, 1928
linear-anisotropic scattering phase function, 438
liquid fraction, 1560
liquid fuel combustion, 844
in non-premixed combustion model, 854
liquidus temperature, 1201
list
drop-down, 30
multiple-selection, 30
single-selection, 30
load balancing, 1606
load distribution
computation, 747
local cell remeshing method, 570
local cylindrical coordinate system, 259
local cylindrical velocities, 266
local face remeshing method, 571
local remeshing
additional, 571
method, 570
log files, 73
logarithmic plots, 1481
loss coefficient, 330
core porosity model, 780
inlet vent, 279
outlet vent, 298
radiator, 333
LS-DYNA,
LS-DYNAFile/Import/LSTC/Input File...,
LS-DYNAFile/Import/LSTC/Input File...File/Import/LSTC/Input File... menu, 80
LSF
checkpointing, 17
job scheduling, 1589
parallel processing, 17
serial processing, 17
LSTC files, importing, 80
lumped parameter models, 324

M

Mach number, 292, 505, 510, 1560
Macro Heat Exchanger Group dialog box, 1677
macros, 777, 1290
magnifying the display, 1427, 1437
Manage Adaption Registers dialog box, 1343, 2129
manuals, using the, lxi
mapping data
electromagnetic losses, 109
fluid-structure interaction (FSI) problems, 113
Marangoni stress, 303, 305
Marc POST files, importing, 81
mass diffusion
to surfaces, 831
mass diffusion coefficients, 439, 446, 818, 1555, 1559
about, 438
full multicomponent, 440
inputs for, 442
kinetic theory, 444
parameterized, 442
using Fick's Law, 439
Mass Diffusion Coefficients dialog box, 443–444, 1773
mass flow inlet boundary, 249, 270
limitations, 271
special considerations, 271

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mass flow rate, 272, 485, 1511
mass flow split, 295
mass flux, 272, 1510
mass fraction, 1561
mass source terms, defining, 246
mass sources, 246
  parameterized, 246
Mass-Flow Inlet dialog box, 271, 1832
Material Properties dialog box, 1748
material properties, checking, 1313
materials, 386
  copying from the database, 389
  creating new, 390
  database, 386, 800, 807
    data sources, 389
    user-defined, 391
  deleting, 391
  discrete phase, 387, 1042, 1054, 1057
  mixture, 387, 800
  modifying, 388
  PDF mixture, 885
  renaming, 388
  reordering, 391
  saving, 390
Materials task page, 387, 427, 488, 1735
matrix size, global, 1624
Maxwell Mapping dialog box, 2066
Maxwell-Stefan equations, 440
mean beam length, 436
mean flow quantities, 1268
mean mixture fraction, 248
Mechanical APDL files
  exporting, 84
    after a calculation, 87
    during a transient calculation, 88
  importing, 81, 153
  mapping data with, 113
Mechanical APDL files, importing, 81, 153
Mechanical APDL Input, 84
Mechanical APDL Input files, exporting
  during a transient calculation, 98
Mechanical APDL Input files, exporting, 84, 88
memory usage, 176
  in multigrid, 1225, 1229
menu bar, 22
menu commands, character strings , 55
Merge Zones dialog box, 184, 2077
merging zones, 184
meridional view, 1449
mesh, 1558
  adaption, 1325
  coarsening
    based on gradient, 1333
    near walls, 1338
  filters
    partition, 1622
    tpoly, 155
  interfaces, 537
    creating, 524, 541
    deleting, 543
    shapes of, 538
  motion of, 513, 535
  moving reference frames, 513
  partitioning, 1603
    automatic, 1604
    check, 1622
    filter, 1622
    guidelines, 1605
    inputs for, 1604, 1606
    interpreting statistics, 1622
    manual, 1605
    methods, 1616
    METIS, 1622
    optimizations, 1620
    report, 1613
    statistics, 1613
    troubleshooting, 1624
    within registers, 1613
    within zones, 1613
  polyhedra
    converting hexcore cells to, 155
    limitations, 1327
  refinement, 244, 1325, 1327
    anisotropic, 1339
    at boundaries, 1328
    based on cell volume, 1337
    based on gradient, 1331
    based on isovalue, 1335
    dynamic, 1333
    in a region, 1336
    near walls, 1338
  replacement, 198
  requirements
    dynamic meshes, 555, 567, 571, 578, 580, 583, 586
    moving reference frame, 515
    multiple reference frames, 524
    rotating flow, 500
    sliding meshes, 540
    swirling flow, 500
    volume of fluid (VOF) model, 1096
  resolution, 1325, 1327
  setup constraints
    dynamic meshes, 555, 567, 571, 578, 580, 583, 586
    moving reference frame, 515

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiple reference frames, 524
rotating flow, 500
sliding meshes, 540
swirling flow, 500
volume of fluid (VOF) model, 1096
setup time, 1325
smoothing, 1349
spacing at walls
in laminar flows, 316
in turbulent flows, 1338
units of, 126
velocity, 1558
Mesh Adaption Controls dialog box, 1347, 2130
Mesh Colors dialog box, 1638
Mesh Display dialog box, 1381, 1635
Mesh Interfaces task page, 1875
Mesh Method Settings dialog box, 1879
mesh morpher/optimizer, 631–632
compass optimizer, 632
introduction, 631
limitations, 631
powell optimizer, 633
process, 631
rosenbrock optimizer, 634
setting up, 634
simplex optimizer, 633
torczon optimizer, 633
Mesh Morpher/Optimizer dialog box, 634, 2088
Mesh Motion dialog box, 627, 1896
mesh motion methods, 551
feature detection, 586
volume mesh update, 586
Mesh Scale Info dialog box, 567, 1882
mesh/check-verbosity, 146, 173
Mesh/Fuse..., 190
Mesh/Fuse... menu, 190
Mesh/Info/Memory Usage menu, 176
Mesh/Info/Partitions menu, 177
Mesh/Info/Size menu, 175
Mesh/Info/Zones menu, 177
Mesh/Merge..., 184
Mesh/Merge... menu, 184
mesh/modify-zones/extrude-face-zone-delta, 181–182, 194
mesh/modify-zones/extrude-face-zone-para, 181–182, 194
mesh/modify-zones/fuse-face-zones, 190–191
mesh/modify-zones/make-periodic, 191
mesh/modify-zones/matching-tolerance, 191
mesh/modify-zones/mrf-to-sliding-mesh, 543
mesh/modify-zones/orient-face-zone, 194
mesh/modify-zones/slit-face-zone, 193
mesh/modify-zones/slit-periodic, 192
Mesh/Polyhedra/Convert Domain menu, 178
Mesh/Polyhedra/Convert Skewed Cells... menu, 182
Mesh/Reorder/Domain menu, 199
Mesh/Reorder/Print Bandwidth menu, 199
Mesh/Reorder/Zones menu, 199
mesh/repair-improve/allow-repair-at-boundaries, 174
mesh/repair-improve/improve-quality,
mesh/repair-improve/include-local-polyhedra-conversion-in-repair, 174
mesh/repair-improve/repair-face-handedness, 174
mesh/repair-improve/repair-face-node-order, 174
mesh/repair-improve/repair-periodic, 174
mesh/repair-improve/repairmesh/repair-improve/improve-quality, 174
Mesh/Replace... menu item, 198
Mesh/Rotate..., 203
Mesh/Rotate... menu, 203
Mesh/Separate/Cells... menu, 188
Mesh/Separate/Faces... menu, 186
Mesh/Smooth/Swap..., 1349
Mesh/Smooth/Swap... menu, 1349
Mesh/Translate..., 202
Mesh/Translate... menu, 202
Mesh/Zone/Activate... menu, 197
Mesh/Zone/Append Case and Data Files..., 157
Mesh/Zone/Append Case and Data Files... menu, 2081
Mesh/Zone/Append Case File..., 157
Mesh/Zone/Append Case File... menu, 2081
Mesh/Zone/Deactivate... menu, 196
Mesh/Zone/Delete... menu, 196
Mesh/Zone/Replace... menu, 195
meshes, 129
accuracy
cell quality, 147
flow-field dependency, 148
smoothness, 148
activating zones, 197
adaption, 147
aspect ratio, 142, 145, 147
C-type, 191
checking, 172
face handedness, 172
face node order, 172
polyhedral cells, 172
quality, 146
choosing type of, 141
colors when displaying, 1383
computational expense, 142
connectivity, face-node, 136
converting to polyhedra, 177–178
deactivating zones, 196

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

deleting zones, 196
displaying, 1379
extrusion, 194
face-node
  hex cells, 140
  polyhedral cells, 141
  pyramidal cells, 140
  quadrilateral cells, 137
  tetrahedral cells, 138
  triangular cells, 137
  wedge cells, 139
files, 62
  reading multiple, 157, 190
filters
  fe2ram, 155
  fl42seg, 149
  tmerge, 159
hexcore, 181–182
importing, 148
  ANSYS CFX files, 154
  ANSYS FIDAP files, 157
  CGNS, 78
  FLUENT 4 files, 156
  FLUENT/ UNS files, 156
  GAMBIT files, 148
  GeoMesh files, 149
  I-DEAS files, 150
  ICEM CFD files, 150
  Mechanical APDL files, 153
  NASTRAN files, 151
  PATRAN files, 152
  preBFC files, 149
  RAMPANT files, 156
  TGrid files, 149
interfaces, 161
  creating, 169
  deleting, 169
  setting up, 200
manipulating, 129
memory usage, 176
modifying zones, 195
multiblock, 157, 190
non-conformal, 161
  algorithm, 167
  requirements, 168
  setting up, 169
O-type, 191
orthogonal quality, 144, 147, 1562
partitioning
  METIS, 83
  report, 177
  statistics, 177
periodic repeats interface option, 164, 169, 543
periodicity
  conformal, 191
  non-conformal, 163
polyhedra
  adaption, 181
  advantages, 177
  converting domain to, 178
  converting skewed cells to, 182
  face-node connectivity, 141
  hanging nodes, 181–182
  limitations, 181–182
quality
  accuracy, 144
  aspect ratio, 145
  element distribution, 146
  improving, 174
  orthogonal quality, 144
  stability, 144
reading, 129, 148
recommendations, 142
reordering
  about, 200
  domain, 199
  zones, 199
repairing, 174
replacing, 198
replacing zones, 195
requirements, 143
  axisymmetric, 143
  non-conformal, 168
  periodic, 143
resolution, 142, 146
rotating, 203
scaling, 200
setup constraints, 143
  axisymmetric, 143
  non-conformal, 168
  periodic, 143
setup time, 141
size, 175
skewness, 142, 147
spacing at walls, 146
statistics, reporting of, 175
topologies, 129
  examples of, 131
  hexahedral, 129
  polyhedral, 129
  quadrilateral, 129
  tetrahedral, 129
  triangular, 129
translation, 202

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

types of, 129
which types to use, 142
zone information, 177
message passing library, 1585
METIS, 83, 1622
Microsoft Job Scheduler, 1597
job scheduling, 1589
MiniVAS Settings dialog box, 2152
Mirror Planes dialog box, 1444, 2002
mirroring the domain, 1441
mixed convection, 697, 1217
inputs for, 698
mixing plane model, 523
enthalpy conservation, 532
fixing the pressure level, 531
model compatibility, 2183
patching values, 1242
pressure boundary conditions in, 528
setup, 528
solution initialization, 1241, 1246
solution procedures, 532
swirl conservation, 532
velocity formulation in, 528
with non-reflecting boundary conditions, 351
Mixing Planes dialog box, 528, 2093
mixing rate, 802, 815, 1101
mixture diffusivity, 486
mixture fraction
boundary conditions, 884
secondary, 1569
variance, 1561
secondary, 1569
mixture materials, 387, 800
creating, 807
PDF, 885
mixture multiphase model
body forces, 1099
droplet diameter, 1146
model compatibility, 2183
patching initial volume fraction, 1189
properties, 1145–1146
solution strategies, 1192
mixture properties, 805, 818
model selections, checking, 1308
Models task page , 1639
modified turbulent viscosity, 1561
modifying zones, 195
molar concentration of species, 1561
mole fraction
soot, 1561
species, 1561
molecular heat transfer coefficient, 448, 898
molecular viscosity, 1561
molecular weight, 405–406, 819
Moment Monitor dialog box, 1279, 1921
moment report, 1515
moments
coefficients of, 1278, 1515, 1523
reporting, 1515
momentum accommodation coefficient, 819
momentum source terms, defining, 246
momentum sources, 246
parameterized, 246
momentum thickness Re, 1562
monitoring
drag coefficients, 1278
forces and moments, 1259
coefficients of, 1278
lift coefficients, 1278
moment coefficients, 1278
residuals, 1269
solution convergence, 1268
statistics, 1277
surfaces, 1259, 1283
volume integrals, 1286
Monitors task page , 1913
morpher, 631
introduction, 631
limitations, 631
process, 631
setting up, 634
Moss-Brookes soot formation model, 978
Moss-Brookes-Hall soot formation model, 978
mouse
functions, 1427
GAMBIT-style, 1427
manipulation, 1425, 1427
probe, 1427
Mouse Buttons dialog box, 1428, 2157
moving domains, model compatibility for, 2183
moving mesh, 513
moving reference frame, 522
boundary conditions, 517
constraints, 518
coordinate-system constraints, 515
postprocessing, 520, 1395
pressure boundary conditions, 517
setup, 516
single, 515
solution procedures, 519
velocity formulation , 516, 518
moving reference frames, 513
patching values, 1242
postprocessing, 1395

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution initialization, 1241, 1246
moving walls, 302
MPEG file, 1303, 1457
MSC Marc, 81
mud, 418
multi-stage scheme
controls for, 1237
modifying, 1237
stability, 1223
multiblock meshes, 157, 190
multicomponent diffusion, 440
theory, 440
multicomponent diffusion model, 438
multigrid solver
aggregative, 1228
algebraic (AMG)
inputs for, 1226
coarsening parameters, 1229
default parameters, 1232
flexible cycle, 1226
full (FMG), 1244
convergence strategies, 1245
using, 1244
full-approximation storage (FAS)
creating coarse grid levels for, 1225
inputs for, 1225, 1232
levels, 1225
inputs for
algebraic (AMG), 1226
full-approximation storage (FAS), 1225, 1232
memory usage, 1225, 1229
performance monitoring, 1226
residual reduction tolerance, 1227
selective, 1228
termination criteria, 1227
turning off, 1229, 1232
V cycle, 1226
W cycle, 1226
with parallel solver, 1226
multiphase flows
body forces, 1099
boundary conditions, 1107
compressible, 1144, 1153, 1166
heterogeneous reactions, 1101
inputs for, 1091
cavitation model, 1153
heat transfer in Eulerian model, 1165
mass transfer, 1104
model compatibility, 2183
postprocessing, 1196
dense discrete phase model, 1199
wet steam flow, 1198
reporting flow rates, 1200
solution strategies, 1187
solving wet steam flow, 1195
user-defined scalar transport equations, 492
using the wet steam model, 1178
wet steam model
properties, 1178
UDF, 1179
multiphase model
coupled solution, 1187
Multiphase Model dialog box, 1093, 1642
multiphase species transport, 1100
postprocessing, 1198
multiple graphics windows, 1412
multiple reference frames, 521
boundary conditions, 524
discrete phase in , 522, 1008
for axisymmetric flow, 522
mesh setup, 524
model compatibility, 2183
patching values, 1242
pathlines, 522
postprocessing, 533, 1395, 1525
pressure boundary conditions in, 524
restrictions of, 522
setup, 524
solution initialization, 1241, 1246
solution strategies, 532
steady flow approximation, 522
velocity formulation in, 522, 524
multiple surface reactions model, 836
multiple-selection list, 30

N

N2O density, 1562
naming files, options for, 61
NASTRAN, 84
NASTRAN files
exporting, 84, 95
during a transient calculation, 98
importing, 81
mapping data with, 113
NASTRAN files, 151
natural convection, 697, 1217
high-Rayleigh-number flows, 700
inputs for, 698
modeling, 697
operating density, 700
solution procedure, 698
natural time, 416
navigation pane, 24
navigation pane menu, 1429

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network Configuration dialog box, 2168
New Material Name dialog box, 393, 1750
NH3 density, 1562
NIST real gas model, 451
  limitations, 468
NIST Real Gas Models, 468
NO density, 1562
no-slip condition, 300, 303–304
node values, 1529
  for contours, 1392
  for pathlines, 1409
  for XY plots, 1472
nodes
  display of, 1383, 1425
noise, 988
nomenclature, 2205
non-blocking surfaces, 715
non-condensable gases, 1153
non-conformal interface algorithm, 167
non-conformal meshes, 161
  requirements for, 168
non-equilibrium chemistry
  rich flammability limit, 847
non-gray discrete ordinates (DO) radiation model
  absorption coefficient, 724
  black body emission factor, 722
  emissivity weighting factor, 722
  inputs for, 721
  refractive index, 724
non-gray P-1 radiation model
  absorption coefficient, 724
  black body emission factor, 706
  emissivity weighting factor, 706
  inputs for, 706
  refractive index, 724
non-gray radiation
  DO/energy coupling, 723
non-gray radiation model
  absorption coefficient, 724
  discrete ordinates (DO) radiation model
    inputs for, 721
  P-1 radiation model
    inputs for, 706
  refractive index, 724
non-isotropic thermal conductivity, user-defined, 236
non-iterative time advancement (NITA), 1259
  inputs, 1219
  supported models, 1221
non-Newtonian fluids, 413
non-Newtonian power law, 415
Non-Newtonian Power Law dialog box, 415, 1755
non-Newtonian viscosity, 413
  Carreau model, 415
  Cross model, 416
  Herschel-Bulkley model, 417
  power law, 415
  temperature dependent, 414
non-premixed combustion model
  ANSYS ANSYS FLUENT solution secondary mixture fraction parameters, 888
  boundary conditions, 883
  condensed phase species, 854
  empirical fuels, 846, 854
  equilibrium chemistry, 839, 843
  flamelet data, 867
  for coal combustion, 844, 853, 855
  for liquid fuels, 844
  fuel inlet temperature, 853
  look-up tables, 871, 874
  model compatibility, 2183
  non-adiabatic form, 844
  non-equilibrium chemistry
    rich flammability limit, 847
  postprocessing, 889
  problem setup, 840
  secondary stream, 845
    solution parameters, 888
  solution parameters, 871
  solving, 887
  species selection, 853
  stability issues, 874
non-reacting species transport, 837
non-reflecting boundary conditions, 288, 339
  general, 351
  limitations of, 340, 352
  parallel processing with, 351
  target mass flow rate, 288
  turbo-specific, 339
  with mixing plane model, 351
normalization, 1523
  of residuals, 1270–1271, 1273
NOx model, 862, 943
  boundary conditions, 958
  coal, 949
  fuel, 944
  fuel streams, 946
    limitations, 947
  gaseous fuel, 949
  inputs for, 943
  intermediate N2O, 944
  liquid fuel, 949
  model compatibility, 2183
  prompt, 944
  thermal, 944

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

under-relaxation for, 958
user-defined functions for, 948
NOx Model dialog box, 944, 1706
NTU Table dialog box, 1671
ntu-model, 769
features, 769
restrictions, 770
numbering files, 61
numerical beach
open channel, 1133
open channel wave boundary condition, 1133
numerical diffusion, 142, 1211
numerical input
decimal separator, 1
Nusselt number, 1573

O

O-type meshes, 191
objective functions, 631–632, 634
oil-flow pathlines, 1405
one-step soot formation model, 974
online help for the GUI, 41
Online Technical Support
access from the interface, 44
opaque walls, 726
open channel
recommendations, 1130
open channel boundary condition, 1121
limitations, 1125
recommendations, 1126
open channel flow, 1121
limitations, 1125
recommendations, 1126
open channel wave boundary condition, 1127
Open Database dialog box, 1746
Operating Conditions dialog box, 405–406, 450, 506, 698, 1806
operating density, 700
operating pressure, 405–406, 449, 506, 698
floating, 507
operating temperature, 698
optical thickness
energy coupling for non-gray DO radiation, 723
optimizer, 631–632
compass, 632
introduction, 631
limitations, 631
options, 634
powell, 633
process, 631
rosenbrock, 634
setting up, 634
simplex, 633
torczon, 633
Options dialog box, 1883
options, startup, 10
Linux/UNIX, 10
parallel
Linux/UNIX, 1599
Windows, 1595
Windows, 10
Orient Profile, 1810
Orient Profile dialog box, 371
orienting
face zones, 194
pictures, 120
profiles, 371
orthogonal quality, 144, 147, 1562
orthographic, 1438
Orthotropic Conductivity dialog box, 424, 1761
orthotropic thermal conductivity, 424, 426
Orthotropic UDS Diffusivity dialog box, 430
outflow boundary, 249, 293
Outflow dialog box , 1836
outflow gauge pressure, 268
outlet vent boundary, 249, 297
Outlet Vent dialog box, 297, 1838
outline display, 1379, 1384
overlaying graphics, 1411

P

P-1 radiation model
non-gray
absorption coefficient, 724
black body emission factor, 706
emissivity weighting factor, 706
inputs for, 706
refractive index, 724
properties, 435
residuals, 735
solution parameters, 733
P-1 radiation source terms, 248
packed beds, 221, 232
packing limit, 1149–1150, 1158–1159
Page Setup dialog box, 38
Parallel Connectivity dialog box, 1625, 2171
parallel processing, 1585
active cell partition, 1548
architecture, 1585
automatic partitioning, 1604
cell partition, 1551
checkpointing, 17
communication, 1630
compute cluster package (CCP), 1597

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data files, 68
efficiency, 1585
exporting, 84
input/output (I/O) capability, 68
introduction, 1585
limitations, 84, 717
load balancing, 1614
load distribution, 1624
manual partitioning, 1605
multigrid settings for, 1226
network connectivity, 1625
network latency and bandwidth, 1628
on a Linux/UNIX system
using command line options, 1599
on a Windows system
using command line options, 1595
using the FLUENT Launcher, 1587
using the Microsoft Job Scheduler, 1597
partition surfaces, 1357
partitioning, 1585, 1603
troubleshooting, 1624
performance, 1626, 1630
statistics, 1626
recommended procedure, 1585
starting the solver
on Linux/UNIX, 1599
on Windows, 1594
using Linux/UNIX command line options, 1599
using the FLUENT Launcher, 1587
using the Microsoft Job Scheduler, 1597
using Windows command line options, 1595
stored cell partition, 1570
thread control, 1625
using LSF, 17
using SGE, 17
Parallel/Auto Partition..., 1604
parallel/network/path, 2170
Parallel/Network/Show Bandwidth..., 1628
Parallel/Network/Show Connectivity..., 1625
Parallel/Network/Show Latency..., 1628
parallel/partition/set/load-distribution, 1624
Parallel/Partitioning and Load Balancing..., 1606
Parallel/Thread Control..., 1625
Parallel/Timer/Reset menu, 1626
Parallel/Timer/Usage menu, 1626
parameterized quantities
density, 403
energy sources, 247
mass diffusion coefficients, 442
mass sources, 246
momentum sources, 246
porosity, 236
resistance coefficients, 228
specific heat capacity, 433
thermal conductivity, 418
UDS diffusivity, 427
viscosity, 408
parameters, 1509
defining, 210
fuel NOx, 949
gaseous and liquid fuel NOx, 950
input, 210
prompt NOx, 949
solid (coal) fuel NOx, 950
SOx, 963
turbulence, 955, 967
Parameters dialog box, 2041
parameters, deformation, 631, 634
partially premixed combustion model, 905
limitations of, 905
model compatibility, 2183
overview, 905
using, 905
partially premixed flames, 893–894, 905
Participating Boundary Zones dialog box, 1663
particle, 1007
accretion, 1017, 1054
cloud tracking, 1045
erosion, 1017, 1054
incomplete, 1013
initial conditions, 1022
laws
custom, 1042, 1046
radiation, 1063
reference frame, 1021
size distribution, 1022, 1035
trajectory calculations, 1009
trajectory reports, 1072, 1078
alphanumeric reporting, 1081
display of, 1068
PDF tracking, 921
step-by-step track report, 1078
unsteady, 1080
turbulence, 1017
particle data
importing, 1070
particle history data
exporting
steady-state, 97
transient, 98
Particle Summary dialog box, 1086, 2039
particle summary, reporting, 1085
particle surface reactions, 807, 810, 835
catalyst species, 836

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inputs for, 835
particle tracking, 1009
Particle Tracks dialog box, 1068, 1980
partition boundary cell distance, 1562
partition neighbors, 1562, 1616
Partition Surface dialog box, 1357, 2137
partition surfaces, 1357
partitioning, 83, 1551, 1562, 1585, 1603
automatic, 1604
check, 1622
filter, 1622
guidelines, 1605
inputs for, 1604, 1606
manual, 1605
methods, 1616
METIS, 83, 1622
optimizations, 1620
report, 177, 1613
statistics, 177, 1613
interpreting, 1622
troubleshooting, 1624
using dynamic mesh model, 1610
within registers, 1613
within zones, 1613
Partitioning and Load Balancing dialog box, 1606, 2161
partitionsurfaces, 1355
passage loss coefficient, 1490
Patch dialog box, 1242, 1939
patching
field functions, 1242
initial values, 1239, 1242
Path Style Attributes dialog box, 1403, 1979
Pathline Attributes dialog box, 1451, 1999
pathlines, 1355, 1400
accuracy control, 1405
animation of, 1451
coarsening, 1404
colors, 1404
in multiple reference frames, 522
oil-flow, 1405
relative, 1405
reversing, 1404
saving, 1405
twisting, 1403
XY plots along trajectories, 1405
Pathlines dialog box, 1401, 1975
PATRAN, 84
PATRAN files
exporting, 84, 95
during a transient calculation, 98
importing, 82, 152
mapping data with, 113
PDF table adiabatic enthalpy, 1562
PDF Table dialog box, 874, 2145
PDF table heat loss/gain, 1563
perforated plates, 221, 233
periodic boundaries, 143, 172, 191, 321, 1441
creating
conformal, 191
non-conformal, 169
non-conformal, 163
slitting, 192
Periodic Conditions dialog box, 494, 1874
Periodic dialog box, 321, 496, 1842
periodic flows, 493
beta calculation, 496
inputs for density-based solvers, 496
inputs for pressure-based solver, 494
limitations, 493–494
overview, 493
postprocessing, 497
pressure gradient, 497
setting up parameters, 496
periodic heat transfer, 763
constant flux, 764
constant temperature, 764
constraints, 764
inputs for, 765
postprocessing, 768
restrictions, 764
solution strategies, 767
theory, 764
periodic repeats interface option, 164, 169, 543
permeability, 222
perspective, 1438
Peters
flame speed model, 898
phase
defining, 1098
for Eulerian model, 1154
for mixture model, 1145
for VOF model, 1136
granular, 1147, 1155, 1170
interfacial area concentration, 1149, 1158
diameter, 1146, 1154, 1553
drag (mixture), 1151
interaction, 1160
Phase Interaction dialog box, 1138, 1151–1152, 1162, 1791
phase localized compressive scheme, 1141
Phase Properties dialog box, 1100
Phases task page, 1098, 1783
picture
files for animation, 1303, 1457

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

options, 117
saving files, 117
using gray-scale colormap, 1418
Picture Options dialog box, 2155
Piecewise Linear dialog box, 907
Piecewise-Linear Profile dialog box, 400, 1751
Piecewise-Polynomial Profile dialog box, 401, 983, 1752
PISO algorithm, 1216
under-relaxation, 1216
pixelation, 720
planar sector, 538
Plane Surface dialog box, 1364, 1929
plane surfaces, 1355, 1364
using the plane tool, 1367
plane tool, 1366–1367
plasma-enhanced surface reaction modeling, 485
Playback dialog box, 1300, 1990
plot interpolated data, 1475
Plot Interpolated Data dialog box, 2018
Plot Profile Data dialog box, 2017
plot/circum-avg-axial, 1476
plot/circum-avg-radial, 1476
Plot/FFT..., 1503
Plot/File..., 1472
Plot/File... menu, 1472
Plot/Histogram..., 1478
Plot/Histogram... menu, 1478
Plot/Modify Input Signal dialog box, 1504, 2020
Plot/Profiles/Interpolated Data... menu, 1474
Plot/Profiles/Profile Data... menu, 1474
Plot/XY Plot..., 1468
Plot/XY Plot... menu, 1468
PLOT3D files
importing, 82
plots
axes in, 1480
external data, 1472
histogram, 1467, 1478
logarithmic, 1481
residual, 1478
solution, 1468, 1478
titles for, 1473
types of, 1466
XY, 1466, 1468, 1472, 1474, 1478
along pathline trajectories, 1405
axis attributes, 1480
circumferential average, 1475
curve attributes, 1482
file format, 1477
profiles, 1474
turbomachinery, 1499
Plots task page , 2011
Point Surface dialog box, 1359, 1927
point surfaces, 1355, 1359
using the point tool, 1360
point tool, 1359–1360
pollutant formation, 853, 862, 943
decoupled detailed chemistry model, 985
NOx, 862, 943
soot, 973
SOx, 959
polyhedra
adaption, 181
advantages, 177
converting domain to, 178
boundary layer treatment, 178
converting hexcore cells to, 155
converting skewed cells to, 182
exporting, 84
face-node connectivity, 141
limitations, 181–182, 1327
Polynomial Profile dialog box, 399, 1751
porosity, parameterized, 236
porosity, user-defined, 236
porous jump, 337
solar load model, 756
Porous Jump dialog box, 338, 756, 1842
porous media, 221
1D, 337
solar load model, 756
anisotropic inertial resistance, 228
enabling reactions in, 228
equations
Darcy's Law, 223
energy, 224
inertial losses, 223
thermal conductivity, 224
transient scalar, 225
heat transfer in, 224
inertial resistance coefficients, 228
inputs for, 225, 231
laminar flow inputs, 234
limitations of, 221
momentum equations for, 222
moving reference frame, 221
multiphase, 238
packed beds, 232
physical velocity formulation, 237
postprocessing for, 240, 1392, 1409, 1472
power-law model, 222, 235
defining porosity, 236
relative velocity resistance formulation, 228
solution strategies, 240
turbulence in, 225

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDFs
inertial resistance, 228
viscous resistance, 228
velocity formulation, 221, 227, 1209
viscous resistance coefficients, 228
positivity rate limit, 1236
postprocessing, 1379
Fast Fourier Transform (FFT), 1500
FW-H acoustics model, 1000
pollutants, 985
reports, 1487, 1509
turbomachinery, 1483
powell optimizer, 633
power law, 411
index, 417
non-Newtonian, 415
Power Law dialog box, 412, 1755
Prandtl number, 1555, 1561
pre-exponential factor, 812
PreBFC mesh files
structured, 83, 149
unstructured, 63, 149
reading, 63
precision, 1, 1269
preconditioning, 1563
premixed
Peters
constants, 898
Zimont
constants, 898
premixed flames, 893
premixed model
C-equation, 896
G-equation, 896
premixed turbulent combustion, 893
boundary conditions, 899
inputs for, 894
model compatibility, 2183
physical properties, 898
postprocessing, 901
progress variable, 899
restrictions, 894
species concentrations, 894, 903
pressure, 1563
absolute, 449
coefficient, 1523, 1563
drop, 330
heat exchanger, 779
dynamic, 1555
gauge, 449
interpolation schemes, 1212
jump
exhaust fan, 299
fan, 326
intake fan, 281
operating, 449
reduced, 1566
reference, 450, 1189
relative total, 1567
static, 1570
total, 1574
pressure boundary conditions, 255
in mixing plane model, 528
in moving reference frames, 517
in multiple reference frames, 524
pressure far-field boundary, 249, 290
Pressure Far-Field dialog box, 291, 1844
pressure inlet boundary, 249, 255
Pressure Inlet dialog box, 255, 1848
Pressure Outlet dialog box, 282, 1853
pressure outlets, 249, 282
non-reflecting boundary conditions, 288
targeting mass flow rate, 288
pressure-based coupled algorithm, 1217
pressure-based segregated algorithm
pressure-velocity coupling, 1215
pressure-based solver, 1207
coupled, 1217
density interpolation schemes, 1213
frozen flux formulation, 1259
limitations, 1207
non-iterative time advancement (NITA), 1259
porous media velocity formulation, 1209
pressure interpolation schemes, 1212
pressure outlets, 286
residuals, 1269
Pressure-Dependent Reaction dialog box, 813, 1768
pressure-velocity coupling
inputs for, 1215
PRESTO!, 501, 519, 698, 1212
previewing the dynamic mesh, 626
Primary Phase dialog box, 1136, 1784
Print dialog box, 38
Problem Setup task page, 1631
processes, 1625
processes, cleaning up, 18
production of
k, 1564
Profile Options dialog box, 1970
Profile Optionsdialogbox, 1387
profile plotting, 1386
scaling factor, 1389
profiles, 210, 366
file format, 367

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interpolation methods, 366
changing, 368
plotting, 1474
reorienting, 371
types, 366
Profiles dialog box, 368, 1808
profiles, boundary, 210, 366
file format, 367
interpolation methods, 366
changing, 368
reading and writing, 70
reorienting, 371
types, 366
units of, 125
progress variable, 899, 1564
sources, 248
projected surface area, 1518
Projected Surface Areas dialog box, 2030
prompt NOx parameters, 949
prompts
about, 50
booleans, 50
default values for, 53
evaluation of, 53
filenames, 51
lists, 51
numbers, 50
strings, 51
symbols, 51
properties, 385
composition-dependent, 800
database, 386, 800, 807
data sources, 389
user-defined, 391
discrete phase, 1054
for solid materials, 386
listing, 1525
mass diffusion coefficients, 438
mixture, 805, 818
modifying, 388
non-gray model refractive index, 724
radiation, 435
absorption coefficient, 436
defining, 723
non-gray model, 724
refractive index, 438
reporting, 438
scattering coefficient, 437
saving, 390
species, 819
temperature-dependent, 398
solution techniques for, 1318
units for, 125, 398
pseudo transient, 1249
calculation, 1252
inputs for, 1250
solution controls, 1251
pseudo-plastics, 415
PTC Mechanica Design files
importing, 82
study, 83
pull velocity, 1579
inputs for, 1204
pull-down menu, 22
pyramidal cells, 140
pyrolysis, 1061

Q

Q3D Mapping dialog box, 2067
Quadratic of Mixture Fraction dialog box, 907
Quadric Surface dialog box, 1369, 1931
quadric surfaces, 1355, 1369
quadrilateral cells, 137
quadrupoles, 988
quality, mesh, 144
checking, 146
improving, 174
quality-based smoothing, 1351
quantities, data file, 121
Question dialog box, 32

R

radial distribution, 1148, 1158, 1788, 1791
radial equilibrium pressure distribution, 283
radial pull velocity, 1565
radial velocity, 267, 1565
radiation
boundary conditions, 724
radiation boundary temperature correction, 724
radiation heat flux, 1510–1511, 1565
Radiation Model dialog box, 705, 709, 721, 733, 1657
radiation models
about, 704
defining material properties, 723
how to use, 705
procedure for setting up, 705
S2S
boundary conditions, 725
reporting values, 739
radiation properties, 435
refractive index, 438
reporting, 438
radiation temperature, 1565
radiative heat transfer

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

black body temperature, 724
boundary conditions, 724
flow inlets and exits, 724
walls, 725–726
discrete phase, 1016, 1063
discrete transfer radiation model (DTRM)
clustering, 738
ray tracing, 738
emissivity, 724
inputs for, 705
material properties, 723
modeling, 704
non-gray discrete ordinates (DO) model
inputs for, 721
non-gray discrete ordinates (DO) radiation model
absorption coefficient, 724
refractive index, 724
non-gray P-1 radiation model
absorption coefficient, 724
inputs for, 706
refractive index, 724
participation in radiation, 731
properties, 435
reporting, 736
solar calculator, 745
solar load model, 740
solution, 732, 735
surface-to-surface (S2S) radiation model
reporting, 739
turning on, 705
Radiator dialog box, 332, 1857
radiators, 330
radio buttons, 29
RadTherm, 84
RadTherm files, exporting
during a transient calculation, 98
RadTherm files, exporting, 84, 96
rake surfaces, 1355, 1361
using the line tool, 1363
RAMPANT case and data files, 70
RAMPANT case files, 156
RANS turbulence models, 645
RANS/LES Interface dialog box, 1858
raster file, 120
rate of fuel NO, 1564
rate of N2OPath NO, 1564
rate of NO, 1564
rate of nuclei, 1564
rate of prompt NO, 1564
rate of reburn NO, 1564
rate of SNCR NO, 1564
rate of soot, 1564
rate of thermal NO, 1564
rate of user NO, 1565
ray file, 709
ray tracing, 707–708
Rayleigh number, 1255
Rayleigh-number flows, 700
reacting flows, 799
eddy-dissipation model, 802, 1101
finite-rate model, 802, 1101
inputs for, 801
overview of inputs for, 799
partially premixed combustion, 905
pollutant formation in, 943
premixed combustion, 893
solution techniques, 1319
Reaction Mechanisms dialog box, 816, 1771
reaction progress variable, 899
reaction rate, 1565, 1576
reactions
defining, 809
for fuel mixtures, 815
zone-based mechanisms, 816, 820
reversible, 819
Reactions dialog box, 1765
Read Mesh Options dialog box, 62, 2047
reading
boundary conditions, 72
boundary profiles, 70
case files, 63–65
multiple, 157
core porosity model parameters, 781
data files, 63–65
multiple, 157
parallel, 69
DTRM Ray files, 709
files, 57
compressed, 59
shortcuts, 57
using toolbar buttons, 61
GAMBIT files, 63
GeoMesh mesh files, 63
injections, 1040
meshes, 62, 129, 148
multiple, 157, 190
parallel data files, 69
PreBFC unstructured mesh, 63
profiles, boundary, 70
surface meshes, 63, 160
TGrid mesh files, 62
view factors, 720
XY files, 36
real gas models

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aungier-Redlich-Kwong, 451
NIST, 451
limitations, 468
UDRGM, 451
example, 480
using the NIST models, 469
using the UDRGM model, 473
real number entry, 29
realizable k-epsilon model
model compatibility, 2183
reduced pressure, 1566
reduced temperature, 1566
reference frames, 1021
multiple, 521
single, 514
reference pressure
actual location, 451
location, 450, 1189
reference temperature, 819
reference values, 1523
for force and moment coefficient reports, 1278
setting reference zone, 1525
setting, general, 1524
Reference Values dialog box, 1524
Reference Values task page, 533, 547, 1898
reference zone, 533, 547, 1395
reflected IR solar flux, 1566
reflected radiation flux, 1566
reflected visible solar flux, 1566
REFPROP database, 468
refractive index, 438, 1566
non-gray radiation, 438
refrigerant, 451
region adaption, 1336
Region Adaption dialog box, 1336, 2122
registers, 1342
combining, 1343
deleting, 1343, 1345
displaying, 1346
manipulating, 1343
modifying, 1345
patching with, 1239, 1243
types, 1342
relative humidity, 1566
relative length scale, 1567
relative Mach number, 1567
relative total pressure, 1567
relative total temperature, 1567
relative velocity, 517, 520, 524, 528, 533, 547, 1241–1242, 1246, 1395, 1531, 1566, 1568
angle, 1567
relative velocity formulation, 518
in multiple reference frames, 522
relative velocity resistance formulation, 221
remeshing methods, 567
2.5D surface, 582
cell zone, 571
CutCell zone, 579
face region, 577
local, 570
using size functions, 571
local cell, 570
local face, 571
remeshing, additional local, 571
remote execution
hiding graphics windows, 1410
remote shell (Linux/UNIX), 1602
Remote Simulation Facility (RSF) dialog box, 13
Rename dialog box, 2043
rendering options, 1425
reordering
domain, 199
meshes, 200
zones, 199–200
reorienting profiles, 371
repairing meshes, 174
Replace Cell Zone dialog box, 195, 2081
replacing
meshes, 198
zones, 195
Report/Fluxes..., 1511
Report/Fluxes... menu, 1510
Report/Histogram..., 1522
Report/Histogram... menu, 1522
Report/Input Summary..., 1525
report/mass-flow, 1200
Report/Projected Areas... menu, 1518
Report/Reference Values..., 1524
Report/Reference Values... menu, 1523
Report/Result Reports..., 511
Report/Summary... menu, 1525
Report/Surface Integrals..., 1519
Report/Surface Integrals... menu, 1519
report/system/proc-stats, 1526
report/system/sys-stats, 1526
Report/Volume Integrals..., 1521
Report/Volume Integrals... menu, 1521
reporting
adjusting integral quantities, 1509
case settings, 1525
center of pressure, 1515
conventions, 1509
cpu usage, 1526
creating surfaces for, 1355

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data, 1509
drag coefficients, 1515
fluxes through boundaries, 1510
force and moment coefficients, 1281
forces, 1515
heat flux, 1510
histograms, 1522
lift coefficients, 1515
mass flux, 1510
memory usage, 1526
moments and moment coefficients, 1515
options for velocity, 1531
output parameters, 1509
parameters, 1509
projected surface areas, 1518
radiation heat flux, 1510
reference values, 1523
summary reports, 1525
surface integrals, 1519
turbomachinery quantities, 1487
volume integrals, 1521
reporting soot quantities, 984
Reporting Variables dialog box, 1984
Reports task page , 2026
reports, mesh check, 173
resetting data, 1249
residence time, 1073
Residual Monitors dialog box, 1272, 1915
residual reduction rate criteria, 1227
residual smoothing, 1226
residuals, 1568
definition of
density-based solver, 1270
pressure-based solver, 1269
display controls, 1276
display of, 1272
divergence of, 1217
for the discrete ordinates (DO) radiation model, 735
for the discrete transfer radiation model (DTRM), 735
for the P-1 radiation model, 735
for the surface-to-surface (S2S) radiation model, 736
monitoring, 1269
normalization of, 1270–1271, 1273
plotting, 1478
reduction of, 1317
renormalization of, 1273
scaling of, 1269, 1271
XY plots, 1478
resistance coefficients, 228
parameterized, 228
user-defined, 228
Results task page , 1965
reversible reactions, 819
Reynolds number, 1510, 1523, 1551
Reynolds stress model (RSM)
model compatibility, 2183
solution strategies, 683
Reynolds stresses, 1576
Ribbon Attributes dialog box, 1403, 1979
rich flammability limit option, 847
rich limit, 847
Riemann invariants, 293
RMS flow quantities, 1268, 1568
RNG k-epsilon model
model compatibility, 2183
root-mean-square flow quantities, 1268
rosenbrock optimizer, 634
Rosin-Rammler size distribution, 1035
Rosseland radiation model
properties, 435
Rotate Mesh dialog box, 203, 2085
rotating
meshes, 203
objects in a scene, 1449
views, 1427, 1435
rotating flows, 497–498, 519
mesh sensitivity in, 500
moving reference frame, 499
overview, 498
pressure interpolation, 1212
three-dimensional, 498
rotating reference frame
patching values, 1242
postprocessing, 1525
solution initialization, 1241, 1246
rotation axis, 217, 219, 515–516, 524, 528
rothalpy, 1568
round-off error, 1269
RSF, 13
hiding graphics windows, 1410
rsh (Linux/UNIX), 1602
RSM turbulence models, 646
Run Calculation task page, 1247, 1249, 1955

S

S2S Information dialog box, 739, 2159
S2S radiation model
boundary conditions, 725
reporting, 739
setting up, 709
surface cluster
defining automatically, 712
defining manually, 711
setting parameters, 711

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surface clusters
controlling, 711
view factor calculation, 713
view factors
computing, 713–714, 717
reading, 720
setting parameters, 711
Saffman's lift force, 1017
Sample Trajectories dialog box, 1082, 2036
Sampling Options dialog box, 1963
SAS turbulence model, 649
Save Output Parameter dialog box, 2044
Save Picture dialog box, 118, 1988
saving
boundary conditions, 72
case and data files, 65, 183
case files, 64
data files, 64
options for, 121
parallel, 68
dialog box layout, 1432
files, 57
compressed, 59
shortcuts, 57
using toolbar buttons, 61
model settings, 72
parallel data files, 68
pathlines, 1405
EnSight format, 1407
FIELDVIEW format, 1406
GAMBIT format, 1407
picture files, 117
solver settings, 72
window layout, 1432
scalar transport equations , 485
scale (in a GUI dialog box), 31
Scale Mesh dialog box, 200, 1634
Scale-Adaptive Simulation (SAS) turbulence model, 649
Scale-Resolving Simulation (SRS) turbulence models, 648
scaling
meshes, 200
objects in a scene, 1449
vectors, 1395
views, 1435
scattering
coefficient, 437, 723, 1569
phase function, 437
scattering coefficient, 437
constant, 437
scattering phase function
Delta-Eddington, 438
inputs for, 437
isotropic, 438
linear-anisotropic, 438
user-defined, 438
scene description, 1444
Scene Description dialog box, 1411, 1444, 1994
Scheme, 47, 49–50, 55
executing functions in the .fluent file, 123
loading source files, 73
Schmidt number, 446, 818
search pattern, 35
second-order accuracy, 1211
second-order time stepping, 546
secondary mixture fraction, 1569
variance, 1569
Secondary Phase dialog box, 1137, 1146–1147, 1149, 1154–1155, 1158, 1785
secondary stream, 845
solution parameters, 888
secure shell (Linux/UNIX), 1602
segregated algorithm
pressure interpolation schemes, 1212
Select File dialog box, 33, 157
Select Input Parameter dialog box, 1807
selecting files, 33
selective multigrid solver, 1228
semi-transparent walls, 726, 728
Separate Cell Zones dialog box, 188, 2080
Separate Face Zones dialog box, 186, 2078
separated flows, 146
separated fluid regions, 693
separating
cell zones, 185, 188
face zones, 185–186
serial processing
checkpointing, 17
using LSF, 17
using SGE, 17
Set Dual Cell Heat Exchanger dialog box, 1668
Set Injection Properties dialog box, 858, 1039–1040, 2097
Set Multiple Injection Properties dialog box, 1047, 2101
Set Shell Thickness dialog box, 2105
Set Spark Ignition dialog box, 1700
Set Units dialog box, 126, 1637
SGE
checkpointing, 17
job scheduling, 1589
parallel processing, 17
serial processing, 17
shadow nodes, duplicate, 174
shape optimization, 631–632

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

compass optimizer, 632
powell optimizer, 633
rosenbrock optimizer, 634
simplex optimizer, 633
torczon optimizer, 633
shear layers, 146
shear stress, 146, 316, 1549, 1578–1579
laminar flow, 316
shear thinning, 418
shell conduction, 313
about, 701
initialization, 704
limitations, 702
physical treatment, 702
postprocessing, 704
setting up, 200
Shell Conduction Walls dialog box, 2104
shock waves, 146, 1327
postprocessing for, 1392, 1409, 1472
show all menu, 1431
show only console menu, 1431
SIMPLE algorithm, 1216
comparison to SIMPLEC, 1216
simple-effectiveness-model, 769
features, 769
restrictions, 770
SIMPLEC algorithm, 1216
under-relaxation, 1216
simplex optimizer, 633
single moving reference frame, 514–515
single phase flows
scalar equations, 488
UDS transport equations, 488
Single Rate Devolatilization Model dialog box, 1061, 1778
single reference frame, model compatibility , 2183
single-precision solvers, 1, 1269
single-selection list, 30
Site Parameters dialog box, 818, 1772
site species, 816
Six DOF solver, 550, 602
rigid body motion, 602
settings, 601
size functions, local remeshing using, 571
skewed tetrahedral cells, 182
skewness, 142, 147
equiangular, 1550
equivolume, 1550
reported during mesh preview, 627
skin friction coefficient, 1523, 1569
slider, 31
sliding meshes , 535
boundary conditions, 541
cell zone conditions, 541
constraints, 538, 540, 545
contour display, 547
file saving, 543, 545
initial conditions for, 522
mesh interface shapes, 538
mesh requirements, 540
mesh setup, 537
model compatibility, 2183
patching values, 1242
postprocessing, 547, 1395, 1525
rotation speed, 541
setup, 541
solution initialization, 1241, 1246
solution procedure, 545
time step, 545–546
translational, 541
slip wall, 300, 303–304, 319
slitting
face zones, 193
periodic boundaries, 192
slope limiter, 1239
Smooth/Swap Mesh dialog box, 1349, 2135
smoothing, 1349
boundary layer, 561
Laplacian, 1351
quality-based, 1351
skewness-based, 1353
SNCR parameters, 954
solar calculator, 745
inputs/outputs, 745
theory, 746
Solar Calculator dialog box, 1665
solar heat flux, 1569
solar load model
applications, 741
boundary conditions, 753
discrete ordinates (DO) irradiation, 744
boundary conditions, 758
postprocessing, 761
animation, 762
serial solver, 747
setup, 748
graphical user interface, 748
GUI, 748
solar calculator, 745
solar ray tracing, 741
boundary conditions, 754
glazing materials, 742
inputs, 743
shading algorithm, 742

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

text interface commands, 759
TUI
additional commands, 761
adjacent fluid cells, 760
align camera, 760
autoread solar data, 759
autosave solar data, 759
commands, 759
ground reflectivity, 761
quad tree refinement, 760
scattering fraction, 760
user-defined functions (UDFs), 748
solar ray tracing, 741
boundary conditions, 754
glazing materials, 742
inputs, 743
shading algorithm, 742
solid (coal) fuel NOx parameters, 950
Solid dialog box, 218, 1802
solid materials, 219, 386, 830, 835
solid species, 807, 816
solid suspension, 1008
solid zone, 218
solidification and melting, 1201
in a VOF calculation, 1145
inputs for, 1201
postprocessing, 1205
solution procedure, 1205
Solidification and Melting dialog box, 1201, 1730
solids pressure, 1148, 1158
solidus temperature, 1201
solution, 1207
accuracy, 1211
animating, 1296
calculations, 1247, 1249
convergence, 1268
convergence and stability, 1317
executing commands during, 1288
gradient limiters, 1239
histograms, 1478
initialization, 1239, 1242, 1249
inputs, 1207
interpolation, 106
for electromagnetic losses , 109
for fluid-structure interaction (FSI) problems, 113
interrupt of, 1249
limits, 1235
pressure, 509
temperature, 509, 693
monitoring, 1277
non-iterative solver (NITA), 1219
parameters
listing, 1525
non-iterative solver, 1219
pseudo transient, 1250
under-relaxation, 1217
procedure, 1207
process, 1207
pseudo transient, 1250–1251
stability, 1217
techniques
compressible flow, 509
discrete phase, 1063
for convergence monitoring, 1277
for heat transfer, 693
for periodic heat transfer, 767
for porous media, 240
for radiation, 732
for reacting flows, 823, 1319
for swirling or rotating flows, 501
for turbulence, 681
step-by-step, 1318
turning equations on/off, 1318
under-relaxation, 1217
XY plots, 1468
Solution Animation dialog box, 1296, 1953
Solution Controls task page, 511, 1218, 1223, 1225–1226, 1250, 1902
solution data
exporting, 84
after a calculation, 85
during a transient calculation, 98
limitations, 84
mapping, 113
solution file management, 106
Solution Files dialog box, 106, 2062
Solution Initialization task page, 1240, 1937
Solution Limits dialog box, 1235, 1904
Solution Methods task page, 519, 1213, 1217, 1900
Solution Steering dialog box, 1958
Solution task page , 1899
Solution XY Plot dialog box, 1468, 2012
Solution Zones dialog box, 2114
Solve/Animate/Playback..., 1300
Solve/Controls..., 511
Solve/Controls... menu, 2117
Solve/Controls/Multi-Stage... menu, 1237
Solve/Controls/Multigrid..., 1227
Solve/Controls/Solution..., 1220
solve/dpm-update, 1066
solve/initialize/fmg-initialization, 1244
solve/initialize/init-flow-statistics, 684
Solve/Initialize/Initialize... menu, 1239
Solve/Initialize/Patch... menu, 1239

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solve/initialize/set-fmg-initialization, 1244
Solve/Iterate... menu, 1247
Solve/Methods... menu, 2116
Solve/Monitors... menu, 2117
Solve/Monitors/Residual... menu, 1269
Solve/Monitors/Statistic... menu, 1277
Solve/Monitors/Surface... menu, 1283
Solve/Monitors/Volume... menu, 1286
solve/set/expert, 1277
solve/set/multi-stage, 1238
solve/set/stiff-chemistry, 824
solve/set/surface-tension, 1138
solver, 1207
  case check, 1314
  density-based, 1207
    limitations, 1207
  double-precision, 1
  formulation, 1207
  inputs for, 1207
  multigrid
    full (FMG), 1244
    full (FMG), convergence strategies for, 1245
    full (FMG), using the, 1244
  parallel, 1585
  pressure-based, 1207
  saving settings to a file, 72
  segregated
    limitations, 1207
  single-precision, 1
  using, 1207
soot density, 1569
soot model, 973
  boundary conditions, 984
  inputs for, 973
  model compatibility, 2183
  Moss-Brookes, 978
  Moss-Brookes-Hall, 978
  one-step, 974
  two-step, 975
Soot Model dialog box, 974–975, 978, 1717
soot quantities
  reporting, 984
soot, mole fraction of, 1561
Soret diffusion, 442
sound pressure data, 1000
sound pressure level, 991, 1732
sound speed, 1569
source terms
  energy, 247
  in conservation equations, 244
  mass, 246
  momentum, 246
  NOx model, 248
  P-1 radiation, 248
  procedure for defining, 245
  Reynolds stress model, 247
  turbulence, 247
    k-epsilon model, 247
    k-omega model, 247
    Spalart-Allmaras model, 247
  UDS transport equations, 248
  units for, 125
  user-defined scalar, 248
sources
  defining energy
    input parameters, 247
  defining mass
    input parameters, 246
  defining mass, momentum, energy, 244
  defining momentum
    input parameters, 246
  units for, 245
SOx model, 959
  boundary conditions, 970
  formation, 959
  fuel streams, 961
    limitations, 962
  gaseous fuel, 963
  inputs for, 959
  liquid fuel, 963
  solid fuel, 963
  under-relaxation for, 971
  user-defined functions for, 970
SOx Model dialog box, 960, 1713
SOx parameters, 963
  solid fuel, 965
Spalart-Allmaras model
  model compatibility, 2183
Spalart-Allmaras turbulence models, 645
Spark Ignition dialog box, 929, 1700
spark model, 929
species, 387, 800
  adding, 807
  boundary conditions, 314, 822
  bulk, 807–808
  cell zone conditions, 822
  defining, 806
    for fuel mixtures, 815
  deleting, 808
  diffusion, 694
  mass fractions, 822
  molar concentration, 1561
  order of, 807, 809, 830
  properties, 819

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

removing, 808
reordering, 808
sources, 822, 829
transport, 799
inputs for, 801
without reactions, 837
Species dialog box, 806–807, 1763
Species Model dialog box, 829, 840, 895, 1681
species transport
modeling diffusion, 438
multiphase, 1100
postprocessing, 1198
specific dissipation rate, 1570
specific heat capacity, 433, 818–819, 1570
composition-dependent, 435
constant, 434
discrete phase, 1059
kinetic theory, 434
parameterized, 433
temperature-dependent, 434
specific heat ratio, 1570
specularity coefficient, 303–304
speed of sound, 1569
spinodal temperature, 1570
split mass flow, 295
spray modeling
inputs for, 1017
spread parameter, 1035
spring-based smoothing method, 552, 555
SRS turbulence models, 648
ssh (Linux/UNIX), 1602
SST k-epsilon model
model compatibility, 2183
stability, 1216, 1317–1318
under-relaxation and, 1217
stagnation pressure, 505
stagnation temperature, 505
standard k-epsilon model
model compatibility, 2183
standard k-omega model
model compatibility, 2183
standard state enthalpy, 447
standard state entropy, 447
Stanton number, 1573
starting ANSYS FLUENT, 1
on a Linux/UNIX system, 10
on a Windows system, 9
using the Remote Simulation Facility (RSF), 13
starting parallel ANSYS FLUENT
on a Linux/UNIX system, 1599
using command line options, 1599
on a Windows system
using command line options, 1595
using the FLUENT Launcher, 1587
parallel
on a Windows system, 1594
startup options, 10
Linux/UNIX, 10
parallel
Linux/UNIX, 1599
Windows, 1595
Windows, 10
static pressure, 261, 274, 283, 292, 1570
velocity inlet, 268
static temperature, 268, 292, 1570
Statistic Monitors dialog box, 1277, 1917
steady laminar flamelet model
setting up, 847
steady-state dynamic meshes, 628
additional local remeshing, 571
step-by-step solution techniques, 1318
stiff chemistry, 824
stochastic particle tracking, 1044
in coupled calculations, 1066
stoichiometry, 800
Stokes-Cunningham law, 1014
stored cell partition, 1570, 1611
strain rate, 1571
stream function, 1571
streams, fuel
NOx model, 946
limitations, 947
SOx model, 961
limitations, 962
streamwise-periodic flow, 493
stretch factor, 1571
strings , 51
study, PTC Mechanica Design, 83
subgrid dissipation rate, 1571
subgrid filter length, 1571
subgrid turbulent kinetic energy, 1572
subgrid turbulent viscosity, 1572
ratio, 1572
subsonic, 261, 505
subsonic compressible flows, 286
subtest kinetic energy, 1572
supersonic, 261, 505
supersonic compressible flows, 286
surface acoustic power, 1572
surface area (projected), 1518
surface CHEMKIN files, 832
surface clusters
S2S radiation model, 711
defining automatically, 712

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defining manually, 711
surface coverage, 1572
surface deposition, 831
rate, 1572
surface dpdt RMS, 1572
Surface FSI Mapping dialog box, 113, 2070
surface integrals, 1519
generating report, 1519
Surface Integrals dialog box, 1519, 2031
surface integration, 1519
surface kinetic mechanism files, 832
surface mesh file, 72
surface meshes
reading, 63, 160
Surface Meshes dialog box, 160, 2132
Surface Monitor dialog box, 1923
Surface Monitors dialog box, 1283
surface tension, 305, 1150
inputs for, 1138, 1162
surface tension coefficient, 1153
surface-to-surface (S2S) radiation model
boundary conditions, 725
reporting, 739
residuals, 736
setting up, 709
solution parameters, 734
surface clusters
controlling, 711
defining automatically, 712
defining manually, 711
setting parameters, 711
view factor calculation, 713
view factors
basis, 713
computing, 713–714, 717
method, 714
reading, 720
setting parameters, 711
Surface/Iso-Clip..., 1373
Surface/Iso-Clip... menu, 1373
Surface/Iso-Surface..., 1371
Surface/Iso-Surface... menu, 1371
Surface/Line/Rake..., 1361
Surface/Line/Rake... menu, 1361
Surface/Manage..., 1376
Surface/Manage... menu, 1376
Surface/Partition..., 1357
Surface/Partition... menu, 1357
Surface/Plane..., 1364
Surface/Plane... menu, 1364
Surface/Point..., 1359
Surface/Point... menu, 1359
Surface/Quadric..., 1369
Surface/Quadric... menu, 1369
Surface/Transform..., 1375
Surface/Transform... menu, 1375
Surface/Zone..., 1356
Surface/Zone... menu, 1356
surfaces
clipping, 1373
creating for displaying and reporting, 1355
deleting, 1376
grouping, 1376
isodistancing, 1375
isosurfaces, 1355, 1371
line, 1355, 1361
monitoring, 1283
partition, 1355, 1357
plane, 1355, 1364
point, 1355, 1359
quadric, 1355, 1369
rake, 1355, 1361
renaming, 1376
rotating, 1375
transforming, 1375
translating, 1375
uses for, 1355
zone, 1355–1356
Surfaces dialog box, 1376, 1936
Sutherland Law dialog box, 410, 1754
swapping, 1349, 1353
Sweep Surface dialog box, 1409, 1984
sweep surfaces, 1409
swelling, 1061
swirl number, 499
swirl pull velocity, 1573
swirl velocity, 267, 498, 1573
for fans, 324, 328
swirling flows, 497–498
mesh sensitivity in, 500
moving reference frame, 499
overview, 498
pressure interpolation, 1212
solution strategies for, 501
swirl velocity, 498
three-dimensional, 498
turbulence modeling in, 499
symbols , 51
symmetry boundary, 319, 1441
Symmetry dialog box, 1859
symmetry in the display, 1441
system commands
about, 53
for Windows, 54

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T

tab (in a GUI dialog box), 28
tangential velocity, 267, 1573
target mass flow rate
  option, 288
  settings for, 288
  solution strategies for, 290
  UDFs for, 290
task page, 26
task page menu, 1430
Tecplot, 84
Tecplot files
  importing, 83
Tecplot files, exporting
  during a transient calculation, 98
Tecplot files, exporting, 84, 96
temperature, 1567, 1570, 1573, 1575, 1578
  reduced, 1566
temperature-dependent properties, 398, 693
  density, 404
  specific heat capacity, 434
  thermal conductivity, 419
  viscosity, 409
termination criteria, 1227
tetrahedral cells, 138
text
  annotation, 1427
  entry, 29
  prompts, 50
  user interface, 47
text interface
  .fluent file, 123
  commands
    abbreviations, 48
    alias, 49
    line history, 48
    scheme, 49
  help system, 55
  interrupts, 53
  menu commands, 55
  menu system, 47
  prompt system, 50
  prompts
    booleans, 50
    default values, 53
    evaluation, 53
    filenames, 51
    lists, 51
    numbers, 50
    strings, 51
    symbols, 51
  system commands, 53
    for UNIX, 53
    for Windows, 54
text user interface, 47
TGrid
  filtering meshes with, 155
  mesh file, 62, 149
thermal accommodation coefficient, 819
thermal boundary conditions
  external heat transfer coefficient, 318
  heat flux, 317
thermal conductivity, 418, 818–819, 1555, 1574
  anisotropic, 422
  biaxial, 423
  composition-dependent, 420
  constant, 419
  cylindrical orthotropic, 426
  discrete phase, 1059
  kinetic theory, 420
  orthotropic, 424
  parameterized, 418
  temperature-dependent, 419
thermal diffusion coefficients, 1574
  about, 442
  inputs for, 445
Thermal Diffusion Coefficients dialog box, 1774
thermal diffusivity, 898
thermal expansion coefficient, 404
thermal mixing, 691
thermal resistance, 311
thermophoretic coefficient, 1059
thermophoretic force, 1016
thin walls, 311
third-body efficiencies, 812
Third-Body Efficiencies dialog box, 812, 1768
Thread Control dialog box, 1625, 2165
threads, 1625
tilde expansion, 60
time step, 1262, 1574
  appending to file name, 65, 100
  discrete phase, 1012
  scale, 1574
time stepping
  adaptive, 1264
  explicit
    restrictions, 1225
  variable, 1266
time-dependent problems, 1255
  adaptive time stepping, 1264
  animations of, 1296
  boundary conditions for, 376
  inputs for, 1256
  mean flow quantities, 1268

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

postprocessing, 1267, 1296
root-mean-square quantities, 1268
solution parameters for, 1262
statistical analysis, 1260, 1268
time-periodic, 546
variable time stepping, 1266
time-periodic flows, 546
titles, 1414, 1473
tmerge, 159
toolbar, 22
graphics, 22
standard, 22, 61
toolbars menu, 1429
toothpaste, 418
topology, global setting, 1500
torczon optimizer, 633
torque, 1491
torque converters, 531–532
total energy, 1574
total enthalpy, 1574
total heat transfer rate, 1512
total pressure, 257, 505, 510, 1574
total sensible heat transfer rate, 1512
total temperature, 257, 273, 505, 510, 1575
tpoly, 155
Trajectory Sample Histograms dialog box, 2038
transcript files, 75
Transform Surface dialog box, 1375, 2139
Transformations dialog box, 1448, 1997
transforming objects in a scene, 1448
transient boundary conditions, 376
transient particles, 1010
Translate Mesh dialog box, 202, 2084
translating
meshes, 202
objects in a scene, 1449
reference frames, 522
views, 1427, 1437
translation velocity, 516, 524, 528
transmitted radiation (IR) solar flux, 1575
transmitted radiation flux, 1575
transmitted visible solar flux, 1575
transonic, 505
transparency, 1447
triangular cells, 137
triangular face approach, 167
troubleshooting of calculations, 1235
tube banks, 221
TUI , 47
commands in .fluent file, 123
Turbo 2D Contours dialog box, 1497, 2176
Turbo Averaged Contours dialog box, 1496, 2175
Turbo Averaged XY Plot dialog box, 1499, 2178
Turbo Options dialog box, 1500, 2179
Turbo Report dialog box, 1487, 2173
Turbo Topology dialog box, 1484, 2094
turbo-specific non-reflecting boundary conditions
overview of, 339
theory, 342
using, 350
Turbo/2D Contours..., 1497
Turbo/Averaged Contours..., 1496
Turbo/Averaged XY Plot..., 1499
Turbo/Options..., 1500
Turbo/Report..., 1487
turbomachinery, 1375
2D contours, 1497
average flow angles, 1489
average total pressure, 1488
average total temperature, 1489
averaged contours, 1496
axial force, 1491
coordinates, 1500
defining topology, 1484
efficiency, 1491, 1494, 1523
mass flow, 1488
passage loss coefficient, 1490
postprocessing, 1483
quantities, 1488
report, 1487
setting global topology, 1500
swirl number, 1488
topology, 1484
torque, 1491
XY plots, 1499
turbulence, 643, 1017
boundary conditions, 678
computing, 252
input methods, 250
relationships for deriving, 252
choosing a model, 644
DDES model, 674
DES model, 650
Detached Eddy Simulation (DES) model, 650
dissipation rate, 1575
Embedded LES (E-LES) model, 651
grid considerations for, 682
grid resolution RANS models, 647
Grid Resolution SRS model, 651
free shear flows, 651
wall boundary layers, 651
hybrid RANS-LES models, 649
inputs for, 655
intensity, 251, 1575

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

k-ε models, 645
k-ω models, 646
kinetic energy, 1576
laminar-turbulent transition models, 646
Large Eddy Simulation (LES) model, 648
length scale, 251
LES model, 648
mesh considerations for, 1338
model compatibility, 2183
model enhancements, 647
model hierarchy, 654
modeling, 643
options, 673
RANS models, 645
reporting, 685
RSM models, 646
SAS model, 649
Scale-Adaptive Simulation (SAS) model, 649
Scale-Resolving Simulation (SRS) models, 648
solution strategies, 681
sources, 247
Spalart-Allmaras models, 645
SRS models, 648
SRS numerics, 652
convergence control, 654
iterative scheme, 653
spatial discretization, 653
time discretization, 652
transition, 216, 237
troubleshooting, 690
user-defined functions, 677
wall roughness, 306, 678
wall treatment RANS models, 647
turbulence model enhancements, 647
turbulence models
model hierarchy, 654
SRS numerics, 652
convergence control, 654
iterative scheme, 653
spatial discretization, 653
time discretization, 652
turbulence parameters, 955, 967
turbulence source terms, defining, 247
turbulent reaction rate, 1576
turbulent Reynolds number, 1576
turbulent viscosity, 1576
modified, 1561
ratio, 252, 1576
subgrid, 1572
subgrid, 1572
twisting pathlines, 1403
Two Competing Rates Model dialog box, 1061, 1779
two-sided wall, 303, 309, 312
two-step soot formation model, 975

U

UDF
anisotropic diffusivity, 432
emissivity weighting factor, 706, 722
scattering phase function, 438
UDF Library Manager dialog box, 2110
UDFs
compilation environment, 8
UDS
anisotropic diffusivity, 428
boundary conditions, setting, 488
diffusivity, 427
anisotropic, 429
cylindrical orthotropic, 431
orthotropic, 430
equations
diffusivity, 427
isotropic diffusion, 427
isotropic diffusivity, 485
postprocessing, 488
setting up
boundary conditions, 488
diffusivity, 488
flux function , 488
solution controls, 488
source terms, 488
UDS Diffusion Coefficients dialog box, 427, 1775
UDS diffusivity
parameterized, 427
UDS Index, 489
UDS transport equations
about, 485
anisotropic diffusivity, 485
diffusion coefficient, 485
multiphase flows, 486, 492
multiphase mass flux, 486
single phase flow, 485, 488
terms
diffusion, 488
flux, 488
source, 488
transient, 488
theory, 488
unburnt mixture
physical properties, 898
species concentration, 894, 903
under-relaxation
default values, 1219, 1251
discrete phase, 1066

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inputs for, 1217
of density, 824
of energy, 693
of temperature, 694
with PISO, 1216
with SIMPLEC, 1216
under-relaxation of variables
density-based solver, 1225
underhood simulations, automotive, 193, 715
Ungrouped Macro Heat Exchanger dialog box, 1672
uniform distribution
for display, 1361, 1364
units, 125, 1547
conversion factors, 128
custom systems, 126
for custom field functions, 1579
for length, 201
restrictions, 125
unsteady flows, 1255
in multiple reference frames, 522
unsteady laminar flamelet model
setting up, 847
using, 850
zeroing species, 864
User Defined Scalar Sources dialog box, 488
user interface
graphical, 21
text, 47
user-defined anisotropic diffusivity, 432
User-Defined Database Materials dialog box, 1746
user-defined fan model, 359
User-Defined Fan Model dialog box, 360, 2115
User-Defined Function Hooks dialog box, 2110
user-defined functions (UDFs), 210
discrete phase, 1018, 1046
turbulence, 677
units in, 125
update of, 1248
User-Defined Functions dialog box, 1753
user-defined mass flux , 485
user-defined materials database, 391
User-Defined Memory dialog box, 2114
user-defined quantities
non-isotropic thermal conductivity, 236
porosity, 236
resistance coefficients, 228
user-defined real gas model (UDRGM), 451
ideal gas equation example, 480
user-defined scalars
defining, multiphase flow, 492
theory, multiphase flow, 486
theory, single phase flow, 485
User-Defined Scalars dialog box, 488, 492, 2113
users, current, 44
using the manual, lxi

V

V-cycle multigrid, 1226
V-LAN Settings dialog box, 2150
vapor pressure, 1060
vaporization
pressure, 1153
temperature, 858, 1059
variable definitions, 1529
Variable Time Step Settings dialog box, 1962
variable time stepping, 1266
Vector Definitions dialog box, 1399, 1974
vector file, 120
Vector Options dialog box, 1395, 1973
vectors, 1393
colors, 1397–1398
custom, 1398
Vectors dialog box, 1394, 1970
velocity, 1568, 1577
angle, 1577
relative, 1567
axial, 267, 1549
boundary conditions, 266
Cartesian, 1531
for boundary condition inputs, 266
components, 1579
cylindrical, 1531
for boundary condition inputs, 266
fixed values of, 242
local cylindrical, 266
magnitude, 1577
radial, 267, 1565
reporting options, 1531
swirl, 498, 1573
tangential, 267, 1573
Velocity Effectiveness Curve dialog box, 1675
velocity far-field boundary, 268
velocity formulation
in moving reference frames, 516
porous media, 221, 227
velocity inlet boundary, 249, 264
Velocity Inlet dialog box, 265, 1860
velocity vectors, 1393
video, 1458
Video Control dialog box, 2148
view factors
compressed row format (CRF), 717
computing, 709, 717
for large meshes or complex models, 717–718

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hemicube method, 714
inside ANSYS FLUENT, 717
limitation, 717
outside ANSYS FLUENT, 718
ray tracing method, 715
reading, 720
reporting, 739
setting parameters, 711
View Factors and Clustering dialog box, 711, 1660
view last, 1433
View menu, 1428
viewing the application window, 1428
views
centering, 1435
magnifying, 1427, 1437
modifying, 1433
rotating, 1427, 1435
scaling, 1435
translating, 1427, 1437
zooming, 1437
Views dialog box, 1434, 1439, 1441, 2000
views, saving, 1440
Views/Save Layout menu, 1432
virtual polygon approach, 167
viscosity, 408, 818–819, 1555, 1561, 1576
Carreau model, 415
composition-dependent, 412
constant, 409
Cross model, 416
kinetic theory, 412
non-Newtonian, 413
Carreau model, 415
Cross model, 416
Herschel-Bulkley model, 417
power law, 415
parameterized, 408
power law, 411
non-Newtonian, 415
Sutherland's law, 410
temperature-dependent, 409
zero-shear-rate, 416
Viscous dialog box, 506
viscous dissipation, 506, 691
Viscous Model dialog box, 446, 510, 655, 1645
viscous stresses, 1238
visibility, 1446
visualizing results, 1379
volatile fraction, 858, 1060
volume adaption, 1337
Volume Adaption dialog box, 1337, 2124
volume fraction, 1577
in Lagrangian discrete phase model, 1008
Volume FSI Mapping dialog box, 113, 2068
volume integrals
generating report, 1521
monitoring, 1286
Volume Integrals dialog box, 1521, 2034
volume integration, 1521
volume mesh update procedure, 586
Volume Monitor dialog box, 1286, 1925
volume of fluid (VOF) model
coupled level set, 1120
donor-acceptor scheme, 1095
explicit scheme, 1095
geometric reconstruction scheme, 1095
implicit scheme, 1097
interpolation
donor-acceptor scheme, 1095
explicit scheme, 1095
geometric reconstruction scheme, 1095
implicit scheme, 1097
jump adhesion
inputs for, 1141
model compatibility, 2183
numerical beach, 1133
open channel
numerical beach, 1133
recommendations, 1130
open channel flow, 1121
limitations, 1125
recommendations, 1126
open channel wave boundary condition, 1127
phase localized
compressive scheme, 1141
properties, 1136–1137
reporting flow rates, 1200
solidification and melting in, 1145
solution strategies, 1189
steady-state calculations, 1097
surface tension
inputs for, 1138, 1162
time-dependent calculations, 1095, 1097
inputs for, 1143
wall adhesion
inputs for, 1116–1117, 1139, 1162
volumetric species, 802
vortex shedding, 1255
vorticity, 1558, 1577
components, 1579
magnitude, 1577

W

W-cycle multigrid, 1226
wall

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

adhesion
inputs for, 1116, 1139, 1162
boundary conditions, 300
motion, 301
shear, 303
thermal, 309
coupled, 193, 312
interface option, 166, 169, 543
fluxes, 1578
heat flux, 317
Marangoni stress, 303, 305
motion, 301–302, 541
moving, 517
no-slip, 303–304
rotation, 524
roughness, 306, 678
shear stress, 316, 1549, 1578
components, 1579
shell conduction, 313
slip, 303–304
specified shear, 303–304
specularity coefficient, 303–304
temperature, 1578
thickness, 311
translation, 524
two-sided, 303, 309, 312
Wall dialog box, 301–303, 308–309, 1865
wall function heat transfer coefficient, 1578
wall surface reactions, 807, 829
boundary conditions, 315
heat transfer, 831
inputs for, 829
site species, 816
solid species, 816
wall treatment RANS turbulence models, 647
Warning dialog box, 32
for merging zones, 2077
WAVE, 381
wedge cells, 139
weighted-sum-of-gray-gases model (WSGGM), 436
wet steam multiphase model
postprocessing, 1198
properties, 1178
solving, 1195
UDF, 1179
using, 1178
window
embed, 1432
window dumps, 121
window layout, 1432
windows, 36
Windows systems
graphics window features, 37
starting ANSYS FLUENT on, 1594
starting ANSYS FLUENT on, 9
using the Remote Simulation Facility (RSF), 13
wireframe animations, 1425
Workbench
automatically saving files within, 68, 1945
launching ANSYS FLUENT within, 2
using input parameters within, 210, 2042
Working dialog box, 32
workpile, 1087
Write Profile dialog box, 71, 1811
Write Views dialog box, 1440, 2002
WSGGM User Specified dialog box, 436, 1776

X

XML template files, 1598
XY plot files, units in, 125
XY plots, 1466, 1468, 1472, 1474, 1480
along pathline trajectories, 1405
axis attributes, 1480
circumferential average, 1475
curve attributes, 1482
file format, 1477
profiles, 1474
residuals, 1478
turbomachinery, 1499

Y

y*
adaption, 1338
y+, 1578
adaption, 1338
yield stress, 417
Yplus/Ystar Adaption dialog box, 1338, 2126
y∗, 1579

Z

zero-shear-rate viscosity, 416
Zimont
flame speed model, 898
Zone Motion dialog box, 626, 1895
zone motion, specifying, 613
Zone Scale Info dialog box, 614, 619, 621, 1894
Zone Surface dialog box, 1356, 2137
zone surfaces, 1355–1356
zones
activating, 197
boundary, 207
changing name of, 209
changing type of, 207

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

deactivating, 196
deforming, 613, 619
deleting, 196
modifying, 195
moving, 513, 535
reordering, 199–200
replacing, 195
zooming, 1427, 1437

Release 13.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.